Design of Automotive Body Assemblies with Distributed Tasks under Support of Parametric Associative Design (PAD)

Gerhard F. K. Tecklenburg

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Abstract

This investigation identifies how CAD models of typical automotive body assemblies could be defined to allow a continuous optimisation of the number of iterations required for the final design and the number of variants on the basis of Parametric Associative Design (PAD) and how methodologies for the development of surfaces, parts and assemblies of the automotive body can be represented and structured for a multiple re-use in a collaborative environment of concept phase of a Product Evolution (Formation) Process (PEP). The standardisation of optimised processes and methodologies and the enhanced interaction between all parties involved in product development could lead to improve product quality and reduce development time and hence expenses.

The fundamental principles of PAD, the particular methodologies used in automotive body design and the principles of methodical development and design in general are investigated. The role which automotive body engineers play throughout the activities of the PEP is also investigated. The distribution of design work in concept teams of automotive body development and important methodologies for the design of prismatic profile areas is critically analysed.

To address the role and distribution of work, 25 group work projects were carried out in cooperation with the automotive industry. Large assemblies of the automotive bodies were developed. The requirements for distributed design work have been identified and improved. The results of the investigation point towards a file based, well structured administration of a concept design, with a zone based approach. The investigation was extended to the process chain of sections, which are used for development of surfaces, parts and assemblies. Important methods were developed, optimised and validated with regard to an update safe re-use of 3D zone based CAD models instead of 2D sections.

The thesis presents a thorough description of the research undertaken, details the experimental results and provides a comprehensive analysis of them. Finally it proposes a unique methodology to a zone based approach with a clearly defined process chain of sections for an update-safe re-use of design models.

Yea, though I walk through the valley of the shadow of death, I will fear no evil for thou art with me; thy rod and thy staff they comfort me.

Psalm 23/4 of David - King James Bible

Do not go where the way may lead you, but where no way is, and leave a track.

Jean Paul
(Johann Paul Friedrich Richter)
1763-03-21 – 1825-11-14
German author

List of Content

List of Tables

List of Figures

Glossary

2D / 3D

2D refers to two dimensions, e.g. height and width of a piece of paper. In manual design 2D methodologies of e.g. descriptive geometry are used to develop 3D parts and assemblies. 3D refers to three dimensions, e.g. height, width and depth of a cube. Under support of a Cartesian coordinate system any point of an object can be described in the 3D modelling space.

Before CAD was established in the 1980th the manual 2D design was standard for the development of automotive body parts. To develop the spatial sense of engineering students the methodologies of 2D design are still used in lecturing at universities.

The CAD system CATIA offers for 2D modelling the work bench *Sketcher* and the 2D application *Work on Support* in the 3D environment. The 2D modelling of sections is the basis of several methodologies for the development of automotive body parts. The standard is to develop 3D geometries under support of 2D sections. After completion of detailing 2D drawings may be produced showing the 3D part or assembly in 2D views, sections and details.

A-pillar (B-, C-, D-pillar)

“Pillars are the vertical supports of the greenhouse of a closed automobile body. They are referred to with letters, such as "A-pillar", "B-pillar" and so forth, moving from the front of the windshield to the rear of the car” (Wikipedia, 2010). Pillars connect the structural parts of the roof rails with the structural parts in the floor area. Cabriolets only have lower B and C-pillars, coupes only lower B-pillars while station wagon, 5-door hatchbacks or SUVs also have D-pillars.

Assembly Constraints

Assembly Constraints (3D Geometric Constraints) are constraints for the assembly (e.g. coincidence of axes or planes) or definition of a relative position (e.g. offset) of parts designed independently. The assembly method using assembly constraints is called *Bottom-Up-Method*. The most robust way to use assembly constraints is, to only apply the constraints to identical published positioning (assembly) geometries such as points, lines or planes defined on both parts.

Bedding

Beddings are functional and geometrical profile areas where the mating flanges of BIW are joint e.g. by spot welding, and weather strips or bonding material are applied for sealing or mounting of other mating parts such as door or wind screen and interior trim panels (door bedding, window bedding etc.). In many cases these profile areas are prismatic (of constant cross section). Special design methodologies are used to design bedding profiles under support of references.

CCP-Link

The CCP Link (Cut Copy Paste Link) is the CATIA definition for a Reference-to-Reference link. This link is defined when two parts are opened in two separate windows outside the assembly context. The link points from the receiver to the sender. The sender does not know the receiver but the receiver knows the sender. The position matrix of the sender is not noticed by the receiver. When the reference part (sender) is moved, the dependant part (receiver) will not follow the movement. Conversely the dependant part (receiver) can be moved independently from the reference (sender).

This link type depends on the element name and the UUID identifiers of the involved documents and is only active, when the documents with the correct UUID, used for the link definition, are loaded.

Class-A

All surfaces of an automotive body exterior and interior which are visible for the customers must be designed with a special attention for modelling quality, curve and surface continuities. These surfaces represent the formal styling of a carline. When CAD-technologies were established in the 1980^{th} special CAD software was developed on basis of Bezier mathematics (e.g. ICEMSurf). The ICEM-Technology company called their high quality surfaces Class-A. Nowadays OEMs have different standards for visible surfaces. For Volkswagen companies Class-A surfaces are all visible exterior and interior surfaces, even the surfaces of the zones a customer can only see when e.g. a door or lid is opened.

Component

In CATIA V5 a component is a 2D or 3D structuring element which cannot be saved separately but in one of the formats of CATIA, such as CATDrawing or CATProduct.

Context-Link

When links are defined within a product structure Reference-to-Instance links are generated.

From the dependant part a link (in CATIA: *Import link* for geometry and *KWE link* for parameters) points to the references on the next higher product level. Conversely independent from the number of *Instance* or *KWE links* one *Context link* points from product level to the dependant part. The dependant part knows its product and the product knows its dependant part. When references are replaced or altered the design of the dependant part automatically will be updated. The Reference-to-Reference link considers the position matrix. As soon as the reference part will change its position the position of the dependant part will be altered too. Conversely the dependant part cannot change its position independently.

Concept section

Concept sections are true most (radial) sections used for the modelling of automotive body parts. In parametric associative design these sections are completely constrained cross sections, prepared to control the constraints of two or more true (radial compound) sections used to design profile areas (sub areas) of the cross section. These true sections are the basis for the derivation of surfaces.

This methodology only works when all areas of responsibility share the concept development of complete zones (zone based approach). If the development of one cross section / zone must take place in different centres of competence control sections are necessary to control the cross section and further more DMU to control the whole 3D zone.

Concept sections and control sections are often called Master sections.

Control section

Control sections are necessary to examine the completed design of the zones of an automotive body when the development of every single zone took place in different centres of competence (see Concept-sections). Control sections are also defined in true most (radial) position.

Concept sections and control sections are often called Master sections.

File-based

The file based data management resigns the use of a product or team data management environment (PDM or TDM).

This method and workflow is possible for the management of relational design projects with distributed tasks for teams with not more than 10 designers when the read and write permissions, the part numbers (names), the instance names the reference names and the development stages (versions) are clearly appointed.

The file based data management requires a team leader responsible for the team data, a project directory for the complete design and a directory for the area design of every single team member. The team leader has the permission to read and write in all directories of the team. The single team member has the permission to read from the project directory and read and write from his own directory. For the overall update of the project directory (project stages, versions) deadlines are appointed where the team leader reads the files from the directories of the team members, checks the observation of the requirements and replaces the files in the project directory.

Feature

In CAD technology the term describes a limited design sequence with the target to complete a design with a special attribute such as a swept surface, fillet, birds beak, hole etc. The design sequence can be an integrated part of a CAD programme or can be defined individually by a designer. CAD programmes allow the storage, cataloguing and reuse of design sequences defined individually. In CATIA features for the reuse of design sequences are called e.g. Powercopy or UDF (User Defined Features).

The combination of several features forms a *master model* for the reuse of the design for a large assembly or design principle. The hypernym for both, *feature* and *master model* is *template*.

Geometrical Set

Geometrical Set (GeoSet, GS) is a folder system (no geometry) used in the CATIA design work benches for wire frame and surfaces. Geometrical sets enable to gather various design features in one set or sub-sets. In Geometrical Sets any feature can be used any number of times. The order of features is no matter but they maintain the relations to their input.

Geometrical Sets allow the individual sorting of features according to design or assembly sequences. GeoSets are used to define traceable and legible hierarchical tree structures. The name of a Geometrical set and its last element (feature) should always have the same name.

Glass house

Glass house is the term used in automotive body engineering to describe all area above the waist line of a car. In Germany instead of "glass house" the English term "green house" is used.

Grey Zone Surfaces

The term "Class A" defines visible (freeform) surfaces in the exterior and interior of vehicles. Class A surfaces show a minimum continuity of *G2 (Curvature Continuity)* or better *G3 (Curvature Change Continuity)*. The design of these surfaces is preceded on base of scanned styling models under support of programmes like ICEMSurf or ALIAS.

Class A surfaces which become visible only after the opening of e.g. a door or lid are defined as *Grey Zone Surfaces*. While the visible Class A surfaces are designed on base of scanned styling models, grey zone surfaces are designed as functional surfaces during body development process first and redesigned as Class A surfaces in a later stage of the design process.

G0-, G1-, G2-, G3-Continuity

Continuity is a mathematical indication of the smoothness of the flow between two curves or surfaces.

G0: Positional continuity: Curves meet or intersect under any angle exactly at their end points, surfaces along their corresponding edges. Positional continuity (G0) means that there is a bend between the curves or a bend in the joined surfaces.

G1: Tangent continuity: Curves meet or intersect exactly at their end points, surfaces along their corresponding edges. Tangent continuity means that there is no bend between the two curves or surfaces but a smooth transition. Both tangents at the corresponding ends of the two curves or surfaces are the same. A simple example for G1-continuity is the fillet between two lines or between to planar surfaces.

G2: Curvature continuity: Curves meet or intersect exactly at their end points, surfaces along their corresponding edges. Curvature continuity means that the two meeting/intersecting curves or surfaces have the same end tangents as well as the same curvature on both corresponding ends.
For the definition of spine curves for surfaces (such as swept surfaces) G2-continuity is expected.

G3/G4: higher continuities: G3-continuity means that not only the conditions of G2-continuity are fulfilled but also that the rate of change of the curvature on both sides of curves or surfaces starting from their meeting point or edge is the same.
For Class-A surfaces (visible design surfaces of exterior and interior) G3-continuity is expected. G4-continuity means that the rate of change of the rate of change of the curvature is the same on both sides.

Import-Link

CATIA definition for a Reference-to-Instance link on base of reference geometry. For detailed description see *Context-Link.*

KWE-Link

CATIA definition for a Reference-to-Instance link on base of reference parameter. For detailed description see *Context-Link.*

Kinematics Constraints

Cinematic is the knowledge about movement without consideration of forces.

The degree of freedom (DOF) of a body depends on the number of possible independent movements the body can perform relative to another body. E.g. a cube can be moved on the planar surface of a second body in two directions and can be rotated around its centre of gravity (DOF=3).

In the design of assemblies some movements are desired while other possible movements are excluded by the design of the single parts. A cinematic is definite when all desired movements are defined by Kinematics Constraints and the degree of freedom is equal zero.

The multitude of possible joints and movements requires a high number of Kinematics Constraints. E.g. a revolute joint has one rotational degree of freedom, a prismatic joint has one translational degree of freedom, a cylindrical joint has one rotational and one translational degree of freedom or a point/curve joint has three rotational and one translational (tangential) degrees of freedom.

Knowledge ware

Knowledge ware (Knowledgeware) is a term used for software for the application of Knowledge Based Engineering / Knowledge Based Design (KBE).

KBE is the integration of expert knowledge, rules and or processes into CAD designs. Expert knowledge collected once can be customised to all people involved and keeps well preserved even when know-how carrier change their position or leave the company. “An efficient KBE environment will release design engineers from time consuming routine work and guaranties standards for process traceability. The single design engineer thereby gets more time for new, ambitious work which requires his creativity. This leads to a higher motivation and effectiveness of every single designer.” (Schneider 2005)

Link

Parametric Associative Designs (PAD) with their link network between design elements only become replicable and alterable structured CAD models by the different kind of links. The choice of link and the point of relationship is an important structuring feature of PAD models.

Important links are Parent/Child relations, Instant-Links (References in an assembly from multiple used elements to the document of origin, Instance-to-Instance-links (Assembly Constraints), Reference-to-Instance-links, defined within one product structure) and Reference-to-Reference links defined between documents opened in different windows.

Master model

The goal of any Parametric Associative Design (*PAD*) should be the deployment of one or more CAD models of assemblies or design principles which can be automatically adapted to new specifications by change or replacement of references. The inner consistency of these CAD models should always deliver correct results with every useful change or adaptation. These so called master models are the results of systematic structured designs build up from a large number of individual or existing *features*.

Master models and *features* are also defined as *templates*.

(In *DMU* context a master model is a collection of 3D data from CAD and other CAE programmes to build a digital prototype.)

Master section

Master section is a common name for Concept sections used for the development of surfaces as well as for Control sections used for the examination of completely designed zones (see Concept section, Control section).

Method

Duden Volume 10(2002) describes „method“ as the mode of execution; the way to reach a defined goal and mentions the similar terms instrument, operation, mode, way, technology.

DWDS (2008) describes “method” as a sytem of rules or systematic arrangement which enables to systematically achieve or display scientific findings or organize practical activities in a rationell way.

In PAD a method can be to structure the design of a large package area into zones or to structure the design of the zones according to the IDO (Input-Design-Output) principle.

Package section

In the early phases of PEP the package process optimises the space allocation for the various parts. To describe these space allocation cross-sections, previously defined section developments and competitors’ data are used. These sections are often distorted in terms of their geometric integrity and therefore can not be utilised for dimensioning and surface derivation and extraction. Package sections are often used to describe the number of parts situated in a zone of the body. They are also used to control certain functions or legal requirements.

Part

Merriam Webster (2007:2) defines the term “part” besides others as one of the often indefinite or unequal subdivisions into which something is or is regarded as divided and which together constitute the whole or as an essential portion or integral element. In mechanical engineering a part is a component used in a higher-level mechanical assembly.

CATIA V5 makes a data format CATPart available which is mainly provided to store the design of a single component of an assembly. In the concept phase of a body development only primary and secondary surfaces are necessary and the data size is more or less small. In this phase it is also useful to define assemblies in CATPart models and use geometrical sets to structure sub assemblies and parts according to the Bill Of Materials (BOM).

Powercopy

Powercopies are templates used in CATIA for the reproduction of standard design principles (e.g. design of a stiffening swage in a sheet metal part, design of surface bands from concept section etc. See examples in appendix). Designed in a separated CAD-model the template can be inserted into another CAD-model as often as necessary (e.g. 35 swages for a floor panel developed by 35 insertions of one powercopy). After insertion powercopies can fully be edited. A similar template is the UDF (User Defined Feature). In contrast to powercopies the UDFs cannot be edited after insertion. Therefore these templates are used for applications third users are not allowed to change.

Product

CATIA V5 defines a data format CATProduct and speaks about product while a platform for the assembly of the single parts is meant. So the product structure in CATIA represents the structure for the assembly of the various sub assemblies and parts according to the BOM.

Publication

In CATIA publication is a method to highlight reference geometries and parameters in the hierarchical tree of an assembly which shall be used for the definition of assembly constraints or MML. CAD-programmes can be initialised to only use published references for the definition of links.

Haslauer (2005) describes publication as a way of access to packed references with unknown object types. As a result it seems to be theoretically possible to e.g. exchange published curves with published surfaces. But the automatic synchronisation of links after the replacement of published references is only possible when the replaced and the replacing reference have the identical name und object type.

Package

“The package manages and harmonises the requirements (demands) for the constructed spaces (summarised complex spaces such as engine compartment, interior, under floor, boot), the ergonomics and whole features of a vehicle in the cooperation with all centres of competence and accompanies the vehicle development from the idea up to end of production. The management of the whole vehicle geometry data and actuality of this data in every development stage is the main task of the package “ Grabner & Nothhaft (2006).

In the layout phase before styling a layout of the vehicle is prepared on base of specified targets to safeguard all important dimensions of the vehicle interior and exterior. Therefore all Carry Over Parts (COP) like engines, gearboxes, chassis, climate control unit etc. including their designed spaces and BIW connections are positioned. The positioned COP, first body structures, expected volumes for occupants, boot and tank etc. and legal and company geometric requirements are the Hard Points for the styling department.

During styling phase there are often conflicts between technical and styling targets which require adjustments, corrections and compromises on both sides.

In package phase 2 after styling phase the layout is refined systematically in cooperation with styling and concept design to describe all designed spaces by contours, surfaces and principle cross sections. The detailed layout confirms the compliance with ergonomic requirements, such as seating and courses of motion, direct and indirect vision of driver, as well as all necessary legal and company requirements. This detailed layout is the basis for the development of derivates and successors.

PAD

The Parametric Associative Design (PAD) of a component or an assembly is the method to control components by parameters, such as numerical values or geometrical elements (curves, lines, surfaces etc.), and to associate e.g. geometry, parts or process steps under support of links. In the context of this work PAD can as well define a single part or assembly design arranged according to the method mentioned above.

RPS-System

RPS is a system of dimensional accuracy points in a part or a welding assembly used for a clearly defined position in design space, for fixation on mounting or measurement devices. If a clearly defined mounting is to be defined all the degrees of freedom in direction X, Y and Z as well as rotation around X, Y and Z axes have to be controlled under support of contact surfaces, position, shape and size of fixing holes.

Strategy

In context with the planning of PAD-models “strategy” defines the way to define and structur geometry along the development process chain. The term „strategy“ must be seen in context with the terms “system”, „method“ and „tactic“.

System

“System“ is an 'integrated whole' of elements which are related to each other and can be seen as united elements bound together by goal, function or purpose and in this context are restricted from their environment (e.g. system for the concept development of an A-pillar, structured into two zones and three transition areas (joints) to mating body structures). Systems are organized and preserved by structures. Structure is the pattern of the system elements and their relations defining a system and its functions (Structuring elements in CATIA are the hierarchical tree, CATProducts and CATParts (data formats for assemblies or parts), Geometrical Sets (folders) and the design features).

 The function of a system is the ability to transfer defined input parameters into defined output parameters. Input and output parameters are the relations of a system to its environment.

 Each system consists of elements (components and subsystems e.g. zone of lower A-pillar, joint between upper A-pillar and roof rails and their different panels for BIW, interior and exterior trim and mounting parts) which are related to each other. Often this relation is a two way interaction.

 A system of this kind can be restricted from its environment (other systems) and divided into subsystems by defined boundaries. In this way the function of a single system can be observed and reflected easily. This restriction is necessary as the human comprehension is restricted to understand the overall functions of large systems (Wikipedia, 2008:1).

Tactic

The terms “strategy” and “tactic” should be handled in a direct context. Strategy normally is used to achieve a superior goal while tactic defines the steps and actions to achieve an intermediate goal. In PAD e.g. the strategy can be to use the modeling method (surface, volume…) in relation to the manufacturing method (deep drawing, moulding…) and the tactic can be to use only update safe functions within a design feature.

Template

In CAD technology templates are the sum of all digital design models for the reuse of standard design concepts. Individual or standard design sequences, so called “features” and extensive structured design models for the reproduction of standard assemblies and standard design principles, so called “master models”, define a large variety of templates.
Templates retain approved design methods and solutions and allow fast partly automated design steps. Templates enable standardisation and reuse of common design concepts. This allows the early approval of design variants and a higher degree of ripeness and higher development quality in early project phases (Haasis *et al*, 2006).

True section

True sections are defined perpendicular (compound radial) to the spine and guide curve. These sections are used for modelling of prismatic profile areas (sub zones). The constraints of the true section are controlled by a concept section which controls a complete cross section with two or more prismatic profile areas. True sections are the basis for the derivation of surfaces (see Concept section).

UDF

UDFs (User Defined Features) are templates used in CATIA for the reproduction of standard design principles (see Powercopy, Template)

UUID

“The Unique Universal Identifier (UUID) is an internal mark, which is defined when creating a CATIA document. The UUID is important for Reference-to-Reference link (CCP link). UUID, file name and time stamp are influencing storing, inserting and replacing of components within a product structure.” (Braß, 2005)

List of Abbreviations

A8	Car line of AUDI
ABC	Substitute for user name in start model, first character first name plus first two characters of family name
ADA	Adapter, file definition used for references in start model
ADP	Adapter, file definition used for references in start model
AK 4.6	Collaborative work group of the five German automotive OEM's regarding CAD/CAM activities and regulations
AUDI	Latin translation of "Horch", one of the roots of the German automotive OEM
ASSY	Assembly
BIW	Body In White
BMW	Bayerische Motoren Werke, German automotive OEM
BOM	Bill Of Materials (Parts List)
BRep	Boundary Representation
CAD	Computer Aided Development/Design
CAE	Computer Aided Engineering (Calculation/Simulation)
CAO	Computer Aided Optimisation
CAPP	Computer Aided Process Planning
CAT	Computer Aided Tolerancing
CATScript	Programming language used in CATIA
CATPart	Data format for a part in CATIA
CATProduct	Data format for an assembly in CATIA
CAx	Computer Aided "Anything"
CATIA	Computer Aided Three-Dimensional Interactive Application (CAD Software of Dassault Systèmes)
CCP	Cut Copy Paste Link

CEG	CATIA Einsatz Gruppe (Collaborative sub work group of AK 4.6 of the five German automotive OEM's regarding common use and timing of CATIA releases)
CFD	Computational Fluid Dynamics
COP	Carry Over Part
CSG	Constructive Solid Geometry
CV	Curve, element type used in CATIA
D4	Internal definition for carline A8 of AUDI
DCM	Data Control Model
DEV	Digital Engineering Visualisation
DfX	Design for X (X=suitable for assembly, manufacturing, total cost…)
DMU	Digital Mock-Up
DPT	Digital ProtoTyping
EBOM	Engineering Bill Of Materials
EDM	Engineering Data Management System
EG	Predecessor of EU, Europäische Gemeinschaft (former *EWG Europäische Wirtschaftsgemeinschaft)*
EU	European Union
FEM	Finite-Element-Method
FEM	Finite-Element-Method, file type used for preparation of reference geometries in start model
GCIE	Global Car Manufacturers Information Exchange Group
GFI	Gesellschaft für Ingenieurleistung mbH, German engineering supplier
GEO	Geometry, file definition used for the design (modelling) work in start model
GeoSet	Geometrical Set, file type used in CATIA wire frame and surface design
GSD	Generative Shape Design, Work bench used in CATIA
HAW	Hochschule für Angewandte Wissenschaften (University of Applied Sciences)

HVAC	Heating Ventilation Air Conditioning
IDO	Input Design Output (Structuring method for design process)
JT	Data format for visualisation and data exchange (Siemens – Unigraphics PLM Solution)
KBE	Knowledge Based Engineering
KIN	Kinematics, file definition used for preparation and execution of kinematics simulations in start model
KPR	Konstruktions-Produkt (Design assembly), file definition for assemblies
LHS	Left-Hand Side
LN	Line, element type used in CATIA
MBOM	Manufacturing Bill Of Materials
MBS	Multi-Body System
MIG	Metal Inert Gas welding, gas-shielded welding
MML	Multi Model Link
MS	Microsoft
NC	Numerically Controlled
NVH	Noise + Vibration + Harshness
NX	CAD software of Siemens PLM
OEM	Original Equipment Manufacturer
OHP	Overhead Projector
PAD	Parametric Associative Design
PC	Powercopy, template used in CATIA
PDGS	Former CAD-Software of Ford Motor Company
PDM	Product Data Management System
PEP	Product Evolution (Formation) Process
PhD	Doctor of Philosophy

RHS	Right-Hand Side
PLM	Product Life-Cycle Management
PT	Point, element type used in CATIA
PM	Parameter, element type used in CATIA
PN	Plane, element type used in CATIA
RPS	Referenz-Punkte-System, system of dimensional accuracy points in parts and assemblies
PT	Point, element type used in CATIA
ProE	Pro Engineer; CAD software of Parametric Technology
SAE	(American and international) Society of Automobile Engineers
SF	Surface, element type used in CATIA
SgRP	Seating Reference Point
SUV	Sport Utility Vehicle
SYRKO	Former CAD-Software of Daimler
TDM	Team Data Management system
UDF	User Defined Feature, template used in CATIA
URL	Uniform Resource Locator, e.g. the "address" of a web page on the World Wide Web
UUID	Unique Universal Identifier
VDA	Verband der Automobilindustrie (association of the German automotive industry
VDI	Verein Deutscher Ingenieure (association of German engineers)
VR	Virtual Reality
WRS	Window Regulator System

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Declaration of Non-Plagiarism

I hereby declare that:

1. The research reported in this thesis, except where otherwise indicated, is my original research.

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5. This thesis does not contain text, graphics or tables copied and pasted from the Internet, unless specifically acknowledged, and the source being detailed in the thesis and in the References sections.

Signed: **Gerhard Tecklenburg**

Date: September 15th, 2010

Publications related to this Research

[1] **Tecklenburg, G. (2004)** 'Die parametrisch assoziative Konstruktionsmethodik in der Karosserieentwicklung und deren Auswirkung auf die Hochschullehre.' *VDI-FVT Conference „Entwicklungen im Karosseriebau 2004" Hamburg.* 04-05 May. Düsseldorf: VDI

English title and abstract:

'How the parametric associative design method influences the design education at the university.'

Within Parametric Associative Design (PAD) in addition to the geometric features of a part or product the designer's intention is stored in form of constraints and dependencies between the geometries. The variation or exchange of the driving parameters will change the geometries of the product according to the programmed intentions. Update-Safety, an associative process chain and the re-use of CAD-models for similar designs are the major objectives of PAD.

Associative CAD-models allow for the first time the representation of complex interdependencies of parts, assemblies and process steps within the same model. The clear definition of referencing geometries and parameters and the possibility to exchange these steering references whenever necessary enables process partners to share the development. Industry speaks about concurrent engineering since decades; PAD delivers the tools to implement this method in design engineering.

The implementation of the parametric associative technology leads to more complex CAD-models. The history trees of the design models become more important than the geometric representation. Rules have to be defined to make CAD-models legible for all process partners involved.

The PAD technology has been used in mechanical engineering with its often more simple geometries since several years. In automotive body design OEMs start with this technology now. The change over to this new technology needs an optimization of the design education at universities. The education in design engineering, calculation, simulation and information technology have to close ranks.

[2] **Brockmeyer, H.; Tecklenburg, G. (2005**) ' Entwicklung einer parametrisch – assoziativen CAD-Modellkette für die Prozesskette der Karosserieentwicklung.' *Mobiles*. No. 31. pp. 58-59.

English title and abstract:

'Development of a parametric associative CAD-model chain for the process chain of automotive body development.'

The development or improvement of methods for the parametric associative design of assemblies of automotive bodies in cooperation with German OEMs is one of the major challenges of the Hamburg University of Applied Sciences.

The styling oriented development process of automotive bodies, world wide legal requirements and several other increasing demands lead to a high number of variants and lots of adjustments throughout the design process. The PAD technology offers methods for concurrent design engineering. The early and detailed planning of project variants and possible adjustments (front loading) in combination with re-usable CAD-models will help to reduce costs and time to market.

The publication shows on the example of the concept development of front door and relating body areas how PAD technology could be used in automotive body development.

[3] **Tecklenburg, G. (2005)** 'Entwicklungspotenzial, Modularisierung, Konstruktionsmethodik am Fahrzeugheck'. Opening Speech. *CTI-Conference „Fahr zeugheck und Heckklappe“.* Sindelfingen. 14-15 December.

English title and abstract:

'Development potentials, modular design and design methodology at the rear end of automotive bodies.'

Modern technologies on cars are fascinating engineers in development and production as well as customers. Which are the main design criteria for automotive bodies of passenger cars and lorries conform to appearance, ergonomics, function and manufacturability?

Automotive engineers work in a permanent area of conflict of styling- design and calculation-/simulation-principles. They work at OEMs, engineering or component suppliers. Their responsibilities in the area of automotive body engineering are the project management, design, calculation, testing and preparation for production.

The presentation shows the multiple topics on the rear end of automotive bodies. On the example of a tail gate functions and design methods are explained. On this foundation the chances for the use of parametric associative design technology are illustrated.

[4] **Tecklenburg, G. (2006)** 'Die parametrisch assoziative Konstruktion einer Heckklappe'. Presentation*. CTI-Conference „Fahrzeugheck und Heckklappe“* Munich. 13-14 December.

English title and abstract:

'The parametric associative design of a tail gate.'

Modern technologies on cars are fascinating engineers in development and production as well as customers. Which are the main design criteria for automotive bodies of passenger cars and lorries conform to appearance, ergonomics, function and manufacturability?

Automotive engineers work in a permanent area of conflict of styling- design- and calculation-/simulation-principles. They work at OEMs, engineering or component suppliers. Their responsibilities in the area of automotive body engineering are the project management, design, calculation, testing and preparation for production.

The University of Applied Sciences Hamburg organised in summer semester 2006 a seminar for the parametric associative design of typical assemblies of an automotive body with distributed tasks for the students engaged for the third time. In hands-on-like environment teams of students had the tasks to use, improve and optimize design methods.

On the example of a tailgate and the corresponding body areas the use of parametric associative design with distributed tasks is explained in detail. One of the main focuses was the idea to distribute the design work zone based and not according to the number of parts involved.

[5] **Schumacher, A.; Tecklenburg, G. (2007:1)** 'Cost reduction using modern development methods in the vehicle body design.' Presentation. *25th European Car Body Conference “Strategic questions of car body engineering today and tomorrow”*. Bad Nauheim. 13-14 March.

Abstract:

Two development engineers explain modern development methods for the design and calculation of automotive bodies. After the explanation of the demands for cost and time reductions typical modern methods of parametric design, knowledge based engineering, distribution of work, associative FEM, crash simulation and structural optimisation are presented based on typical examples of body development.

[6] **Tecklenburg, G. (2007:2**) 'Parametric Associative Design of a Tail Gate.' Presentation. *4th Metzeler Sealing Conference “Fahrzeugtüren und -klappen 2007”.* Lindau. 20-21 June.

Abstract:

Modern technologies on cars are fascinating engineers in development and production as well as customers. Which are the main design criteria for automotive bodies of passenger cars and lorries conform to appearance, ergonomics, function and Manufacturability?

Automotive engineers work in a permanent area of conflict of styling, design and calculation/simulation-principles. They work at OEMs, engineering or component suppliers. Their responsibilities in the area of automotive body engineering are the project management, design, calculation, testing and preparation for production.

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On the example of a tailgate and the corresponding body areas the use of parametric associative design with distributed tasks is explained in detail. One of the main focuses was the idea to distribute the design work zone based and not according to the number of parts involved.

[7] **Tecklenburg, G. (2007:3)** 'Karosseriekonstruktion: Parametrisch Assoziatives Konstruieren.' *Mobiles*. No. 33. pp. 14-19.

English title and abstract:

'Automotive body design: Parametric associative design'

The development process of automotive bodies is mainly managed by automotive body engineers. The demands of so called body design engineers in automotive companies have changed a lot throughout the last two decades. Designers became more and more poor managers of design and project work. Project managers with own design experiences get retired. Young project managers do not have enough design experience and sometimes even do not want to design themselves at all.

A good body engineer is only successful when he runs his management mission on base of a broad professional competence such as soft skills and management competence, a good foundation of mechanical engineering knowledge and a broad expertise about the methods used along the process chain of automotive body development.

The complexity and prospects of parametric associative design in the process chain open up to the managing engineer only when he has detailed knowledge for the use of modern CA-tools as well.

[8] **Tecklenburg, G. (2007:4)** 'Die parametrisch assoziative Konstruktion im Entwicklungsprozess Karosserie'. *Conference „Parametrisch assoziative Entwicklung von Baugruppen der Fahrzeugkarosserie - Visionen und Erfahrungen für zukünftige Entwicklungsprozesse".* HdT Essen. 22-23 November. Renningen: Expert

English title and abstract:

'The Parametric Associative Design in the development process chain of auto motive body.'

The fundamentals of parametric associative design (PAD) in automotive body design will be illustrated and discussed. Rules to manage the complexity of PAD-models are pointed out. The process chain "automotive body development" gives an example how PAD can be used to structure and control development processes. Up-to-date examples of associative design in automotive body assemblies validate the definitions.

In conclusion the use of PAD in design engineering education at the university is explained.

[9] **Tecklenburg, G. (2008)** 'The Parametric Associative Design (PAD) of Vehicle Closures.' Presentation. *7th International CTI Forum* '*Automotive Doors*'. Bonn. 12-13 February.

Abstract:

After an introduction regarding the University of Applied Sciences Hamburg and in particular the department automotive and aeronautical engineering with its major 'automotive body development' the situation and interaction between OEMs, suppliers and universities throughout the product development chain are explained.

Parametric Associative Design (PAD) principles enable the process partners involved to share development data and knowledge in clearly defined ways. The presentation gives an insight on fundamentals of PAD such as the different possibilities for associative product models, the ideas of reuse and templates, legibly structured design models, zone based or part based design work. On the example of doors and tailgates the handling of these methods and other door specific methods is illustrated.

[10] **Tecklenburg, G. (2008)** 'Parametrisch assoziative Entwurfsmethoden bei der Modellierung von Baugruppen der Karosserie.' Plenary address. *Conference* '*ICEM - Information Day* ' ICEM Technologies, Hanover. Munich. 21 October.

English title and abstract:

'The Parametric Associative Design methods for the development of typical design spaces of automotive body.'

The associative process chain in the area of concept development of automotive bodies is interrupted in several stages. One reason for an interruption is the interaction between Class A – surfacing and concept design.

The Class A – surfacing process often uses programmes (such as ICEMSurf) which deliver very good options for the Class A design process itself but do not allow the direct interaction on an adequate quality level.

In early phases of concept development the Class A surface models are too late and full of mistakes (such as doubled or damaged surfaces and curves).This leads to a duplication of work as the PAD designer has to define dummy-data himself to fulfil the data quality expected in a PAD design.

The outcome of a conversion of Class A data into parametric data are isolated and often damaged data. The isolation of data makes sense as it is not allowed to change Class A data. On the other hand it takes a lot more effort to repair and slightly change (!?) converted Class A data than to update a well structured PAD model after the exchange of references.

The presentation in front of Class A designers tries to open up the mind of the audience for the challenges of a cooperative design process chain. Examples for the parametric design of gaps, grills, Class B surfaces or textured interior surfaces show that the use of PAD in typical areas of the Class A surfacing process is possible.

[11] **Tecklenburg, G.; Untiedt, S. (2008)** 'Parametrisch assoziative Package-Schnitte in der Prozesskette Karosserieentwicklung. Presentation. *VDI-EKV Conference 'CAD-Produktdaten 'top secret?!' '*. Munich. 02-03 December. Düsseldorf: VDI

English title and abstract:

'The Parametric Associative Package-Sections in the development process chain of automotive body.'

Within Parametric Associative Design (PAD) in addition to the geometric features of a part or product the designer's intention is stored in form of constraints and dependencies between the geometries. The variation or exchange of the driving parameters will change the geometries of the product according to the programmed intentions. Update-Safety, an associative process chain and the re-use of CAD-models for similar designs are the major objectives of PAD.

The associative process chain in the area of concept development of automotive bodies is interrupted in several stages. One reason for an interruption is the reuse of isolated package sections from former product developments. Package sections often taken as results of former designs just describe the parts and assemblies to be used in a design space of a new development. As these sections are often grit sections cut through inclined and curved body areas dimensions of these sections do not allow to use segments for the development of curves and surfaces.

The publication and presentation shows on the example of the inclined and curved upper A-pillar how on basis of Class A – surfaces, curves and reference surfaces a new 3D process chain of (1.) a true most cross section is defined to set parts, sub-assemblies, proportions and dimensions for the design space, (2.) true sections for three different prismatic areas (inner and outer door bedding, window bedding) are defined to develop prismatic surfaces and (3.) offsets of reference surfaces complete the design space. After completion of the 3D design a new true most section and package sections for the examination of e.g. binocular obstruction according to legal and company requirements are cut.

The achievement is a 3D design of the upper A-pillar zone which can be fully re-used and edited for further developments on the same project or similar successors and in different stages of the process chain.

[12] **Tecklenburg G; Haritos G (2009)** Modular System for Methodological Design of Vehicle Body Assemblies. Automob. Tech. Z. *(ATZ* worldwide eMagazines Edition*)* 12: 956-963

English abstract:

The University of Applied Sciences (HAW) Hamburg and the University of Hertfordshire in collaboration with the automotive industry set up a R&D process to optimise the design methods in the “new” parametric associative approach. The objective with the parametric associative design was to develop new methods for task allocation and a more “structured” approach in the organisation of design processes while designing automotive body components. Groups consisting of four to seven students evaluate assemblies for the body. 22 realistic projects were identified before by automotive companies. Here, important conclusions are presented for the example A-pillar.

[13] **Tecklenburg G (2010)** 'Parametrisch assoziativer Ansatz zur Konstruktion von Baugruppen der Karosserie' *Conference PLM Forum 2010.*16-17 June. Mannheim, Dassault Systemes Deutschland GmbH

English title and abstract:

'Parametric associative approach for the design of automotive body asseblies.'

Introduction, portrait of the university, development of design methods in cooperation with the automotive industry, cooperation with software providers ICEM and TransCat PLM, Parametric associative approach for the design of automotive body assemblies: package sections, true most concept section, true (compound radial) sections for the derivation of prismatic surfaces, efficiency analysis of design methods according to criteria applicability, traceability, update safety, time saving by replication.

[14] **Tecklenburg G (2010)** 'Die parametrisch assoziative Konstruktion im Entwicklungsprozess Karosserie. Einführung, Ziele, Ergebnisse'. In: Tecklenburg G (ed) 'Die digitale Produktentwicklung II', 1st edn. Expert, Renningen, pp 1-8

English title and abstract:

'The role of parametric associative design in the Product Evolution Process (PEP) of an automotive body – introduction, goals, results.'

The development or improvement of methods for the parametric associative design (PAD) of assemblies of automotive bodies in cooperation with German OEMs is one of the major challenges of the Hamburg University of Applied Sciences.

The styling oriented development process of automotive bodies, world wide legal requirements and several other increasing demands lead to a high number of variants and lots of adjustments throughout the design process. The PAD technology offers methods for systematic concurrent design engineering. The early and detailed planning of project variants and possible adjustments (front loading) in combination with re-usable CAD-models will help to reduce costs and time to market.

The publication gives an introduction and defines the main goals for the use of PAD technology throughout automotive body development.

[15] **Tecklenburg G, Schubert S, Haritos G (2010)** Design of prismatic areas on assemblies of the automotive body. *International Journal of Advanced Manufacturing Technology* - In peer reviewing process –

English abstract:

During the past few years 25 team projects (with 4 to 8 designers each) in cooperation between Hamburg University of Applied Sciences, University of Hertfordshire and companies of the German automotive industry have been carried out to establish a systematic parametric associative design (PAD) process for assemblies of the automotive body. In context with the product development design methods, methods to structure the information flow within the Computer Aided Design (CAD) - model, the application of Knowledge Based Engineering (KBE) and the distribution of design work for automotive body assemblies have been investigated.

The design work during concept development of an automotive body is characterised by the use of 2D sections and 3D wire frame and surface models. On base of package sections the (radial) concept sections are developed. The concept sections are the reference for true (compound radial) sections which prepare the generating segments for the derivation of surface bands. Surface bands are the basis for the design of body zones, junctions of two ore more body zones, filleting, detailing and the derivation of parts.

The automated reuse of package, concept- (radial) and true most (compound radial) sections for the derivation of surfaces and parts in PAD-models requires design methods which reduce the effort for the creation of the CAD-models, are traceable, easy to use and are update safe.

Most parts of an automotive body feature prismatic areas, especially in the joining areas to corresponding parts. The design of prismatic profiles requires special design methods. In the current research project the design of parametric associative package sections and the special methods for the design of prismatic areas have been evaluated in detail. The present article describes methods for the reliable design of frequently required prismatic surface areas.

[16] **Tecklenburg G, Schubert S, Haritos G (2010)** Konstruktion prismatischer Bereiche an Baugruppen der Fahrzeugkarosserie. *Konstruktion ??.* pp. ?-?.
– In peer reviewing process. Will be published May/June 2011 –

English title and abstract:

'Design of prismatic areas on assemblies of the automotive body.'

During the past years 25 team projects (4 to 8 designers) in cooperation between Hamburg University of Applied Sciences, University of Hertfordshire and companies of the automotive industry have been carried out to establish a systematic parametric associative design (PAD) process for assemblies of the automotive body.

The automated reuse of package, concept- (radial) and true most (compound radial) sections for the derivation of surfaces and parts in PAD-models requires design methods which reduce the effort for the creation of the CAD-models, are traceable, easy to use and are update safe. The present article describes methods for the reliable design of frequently required prismatic surface areas.

Chapter 1 Introduction

One of the main tasks of automotive body design is to define, collect and optimise the methods of car body design. The applied methods of design in the automotive body design sector differ from those applied in mechanical engineering in general.

With introduction of the conventional Computer Aided Design (CAD), the design methods have been relegated to a second place due to the fact that for the task only the final geometry result of the system parts appears to be of importance, and neither the geometry generation nor the controlling parameters and references are given due attention.

During the last decades Original Equipment Manufacturers (OEMs) have changed their design strategy and started to produce more and more different car models in the part of the automotive industry. This involves a rapidly increasing shift of product development, from the OEM to suppliers and engineering partners. Design methods and processes are no longer the central point of interest for the OEM product developers. As a result, the OEMs have lost a great deal of know-how in product development methods.

The Parametric Associative Design (PAD) offers the possibility of representing development methods structured in CAD-models as well linking design, calculation and planning processes with each other. In addition, there is the possibility to store knowledge of these processes in the CAD-models and to reuse them in a “new” product development cycle. An inherent feature of PAD is that in addition to the design of a product, the designer’s intention is stored in form of constraints and dependencies between the geometries and parameters of the various parts. A variation or exchange of the controlling parameters will change the design of the product according to the design intentions. The focus in body design is to define and optimise characteristics and dependencies within parts and assemblies according to the three major requirements in body engineering: styling, functionality and manufacturing.

In contradiction the focus in conventional CAD-design is to develop the geometry of single parts. For optimisation and development parts have to be totally redesigned or modified extensively, whereas PADs can be totally updated, either automatically or in single steps without - or with a minimum of remodelling the geometry of the parts. While the conventional design is only one of many tools within the process chain, the PAD-model assumes a leading role within the process from concept development to manufacturing. Information stored in the parametric design of a body assembly is, for example, the basis for tool and jig design in manufacturing, for tolerance investigations and for quality control.

There are various links defined in PADs at different levels. For example, links between geometry and design features control the geometries at part level; links between parts define shape and position of the parts within an assembly as well as the function of the assembly. Links between design tables and parameters control the values of part parameters, links between a 3D-model and its related drawing, control views of the drawing and assembly constraints define the positioning of related parts.

A body designer who has to satisfy conflicting interests from different partners within the process chain into the CAD-models now assumes a much more important role in the development process than that in the past. The classical distinction between a body engineering designer and body engineering manager is unsuitable for the new parametric design process. The “new” body engineer designs the assemblies and manages the body development process, and he is strongly supported by the parametric design models described. This new design approach needs system engineers with an excellent understanding of how their components relate to associated systems, which can coordinate the development processes and who at the same time are highly proficient CAD-designers.

In the interests of a cost- and time-optimised development process, a collaboration of all parties involved in the product development and its application to production cycle, must and can be redefined with the help of the methods of parametric associative development. This thesis will explore and contribute to this development process.

1.1 Current Situation in Automotive Industry and resulting Tasks for Product Development

In figure 1.1 the conflicting targets for a global acting automotive development are shown. In the past competition in the automotive industry took place between a few manufacturers on a very regional basis.

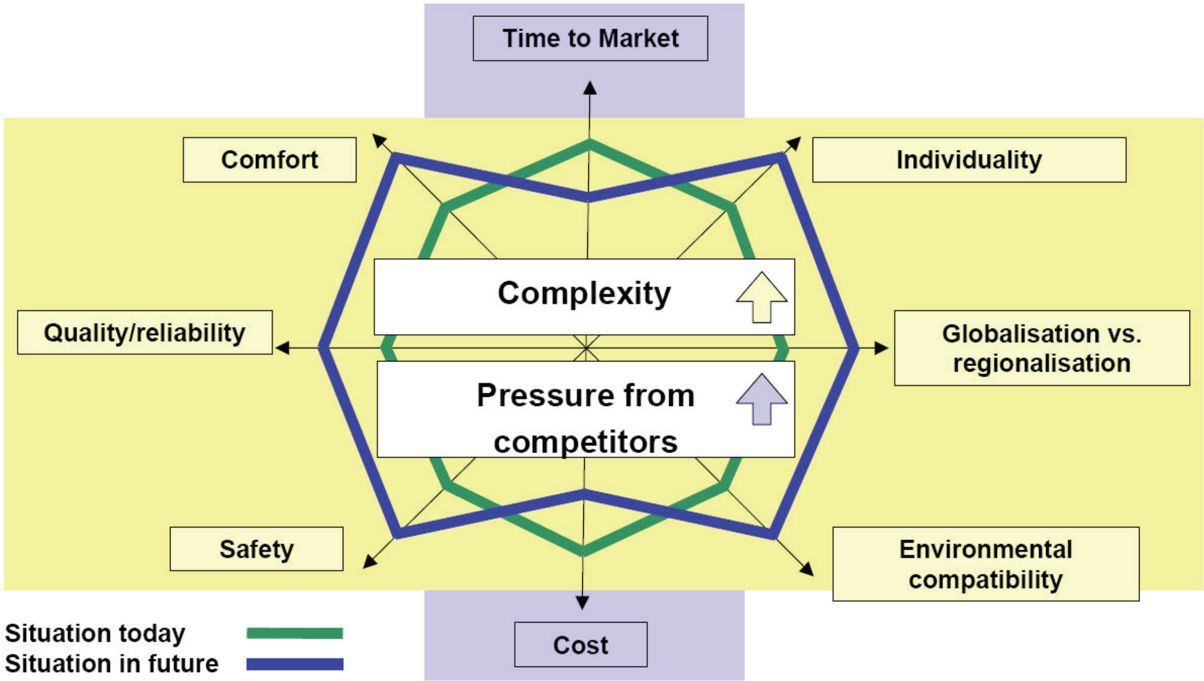


Fig. 1.1 Conflicting targets in the automotive industry caused by external factors (Mantwill, 2007)

Today, however, the competition takes place between a significantly larger number of manufacturers and products on an international level as pointed out by Clark & Fujimoto (1992). According to German association of the automotive industry (VDA, 2006) and Baur (2007) a global competition and an increasing market dynamic during the past years have led to an unprecedented change in design strategy (see figure 1.1). The higher complexity of the vehicles and the model offensive are characterised by an increasingly regional customer orientation and growing demands on the product, which leads to pressure to reduce development costs and time (Bauer 2002), (Mantwill et al 2006).

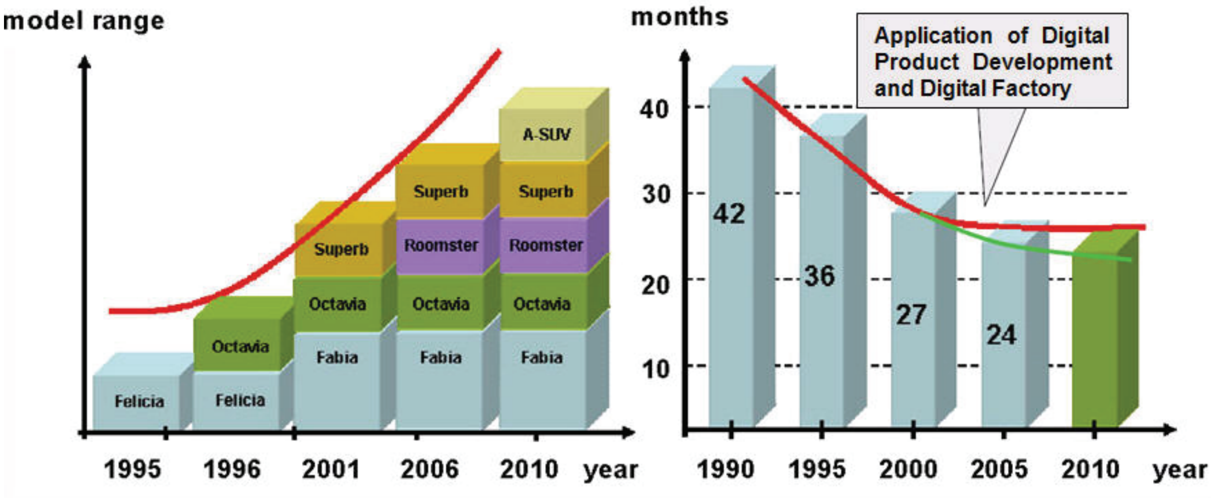


Fig. 1.2 Increase of model range and variants vs. decrease of development time at SKODA Auto a.s. (Baur, 2007)

The development environment for the body design engineer has been steadily changing for the last few decades. During the 50's car bodies were developed on the basis of Class A surface design, principle sections and designs only largely supported by model makers (see chapter 2). In the 70's an almost complete surface description of all design elements of a car body was produced through a large number of designs, which were, still manually produced. All requirements – such as producibility, assembly and occupant ergonomics – were evaluated by means of a large number of physical mock ups and prototypes built. With the introduction of CAD technology the two dimensional design was replaced by the development of 3D design in which the surfaces are completely described.

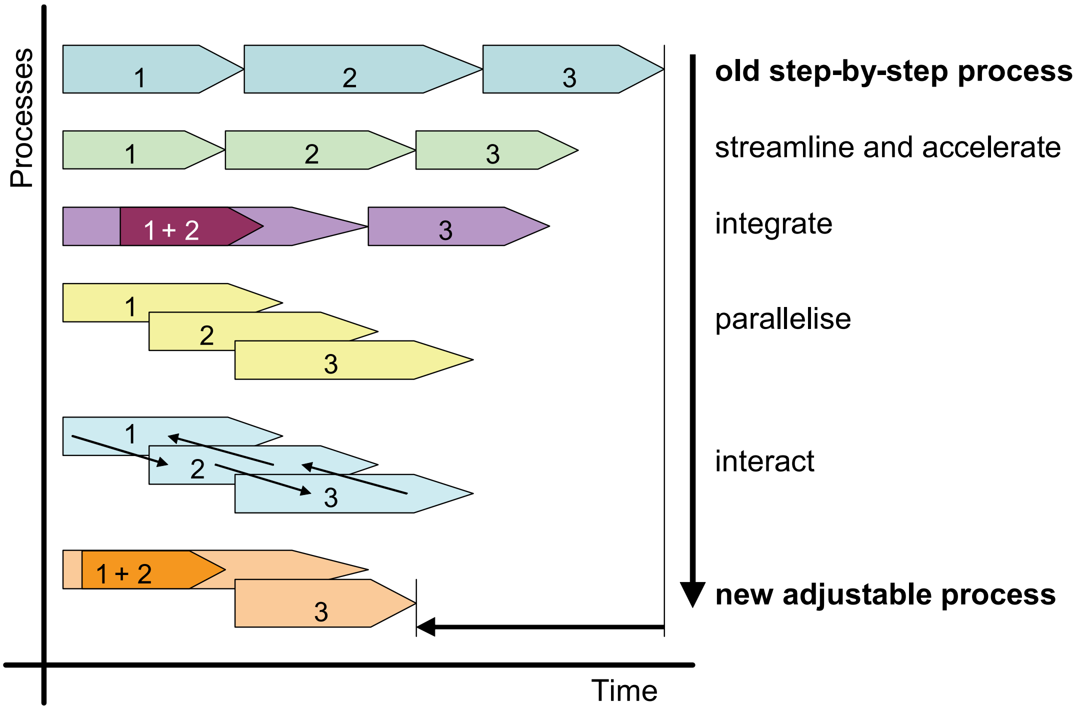
Today the 3D designs are the 3D master for digital or virtual product development. The 3D designs of Class A surfacing and mass production development are thus the basis for geometry formation and determination of model design, which in turn are amplified by Digital Mock Up (DMU) assembly and process simulation and further functional virtual control methods such as, simulations for crash, passenger protection, riding comfort under conditions inducing vibration or fluid dynamics. This, however, does not mean that virtual body design development can replace entirely a real product with simulation. It rather provides the early, consistent and continuous implementation of Computer Aided Anything (CAx) methods in order to ensure that the physical prototypes are able to meet the demands required in shorter time

(Schöneburg, 2007; Breitling, 2007). A further reduction of development time (time to market) and costs is possible by the consistent application of virtual methods in product development and production planning and control (see figure 1.2).

The 3D-CAD model is thus playing an important role as knowledge based system. The conventional 3D-CAD models were purely geometrical models that could not provide the information needed for development. Modern parametric associative 3D CAD models, on the other hand, not only store the history of their development, but can also explicitly acquire further expert knowledge, which can be accessed (e.g. material, weight, centre of gravity, 3D dimensions and tolerances, table of welding spots and related validated weld guns, inclusion of design variants, tables of optimized parameter ranges or standard parameters etc.). The complete or partial reuse of a parametric associative CAD model for product development and further processes results in a reduction of development time and costs and facilitates the capacity for further innovative development (see chapters 3 et seq).

An effective means for reducing development time is to use concurrent engineering developmental procedures (see figure 1.3). The parametric associative design based on explicit references enables subsequent substitutions and adaptations to new reference data. This is the basic principle of the early concurrent/simultaneous engineering of components, which is based on simple self-generated geometry, which is later replaced by the actual reference geometry.

There is, however, a danger that the integration of expert knowledge in a CAD model will lead to a loss of knowledge if the CAD model is passed on thoughtlessly in a global development environment. The rights of the knowledge-based creator and his designs must be protected. However, a smooth design process from layout to mass production development requires the transposition of the reference geometries from design process to design process.



Process 1: Early processes such as Layout, Concept, Styling, Package
Process 2: Intermediate processes like Surfacing, Calculation, Simulation
Process 3: Late processes such as Design Mass Production, Tool Design

Fig. 1.3 Shorter and adjustable process chain

The know-how which underlies knowledge-based designs and is complex in some areas must be constantly updated and adapted to new models, variants, software and hardware environments. The new methods of parametric associative design, which are closely associated with calculation and simulation processes, thus require a comprehensive new organisation of the collaboration between OEMs, suppliers and the engineering partners.

1.2 Possible Approaches

The process chain of body development today produces a lot of redundant data (see chapter 5). Although the methods of parametric associative construction enable abbreviated, adaptable processes, the definition of linked processes at the drafting stage of the vehicle to its production planning fails because of several factors, such as:

a) A continuous use of data is not achieved because departments adhere to software solutions which correlate to each other poorly due to differences in focal points and data format.

b) The transfer of isolated geometry is restricted due to “political” decisions, so that in follow up processes reference geometries must be redesigned. An integration of design and calculation into follow up processes requires, however, a clear definition of the controlling data to be produced and stored additionally in the structural tree by the design engineer.

An example for subject a) is the interface between Class A surfacing and concept design. The design requires that surfacing provides

i. offset-capability (for design with material thickness),
ii. continuously closed surfaces (expected surface quality) with
iii. the first bent portion (expected gap quality),
iv. all related continuous curves (spine/guide curves for design process), and
v. untrimmed surfaces (reference surfaces for design process).

The data which is provided by surfacing after the conversion into a native CAD-format often contains gaps and continuity errors and must be repaired. Untrimmed surfaces do not existent or are duplicated. Class A surfaces are often not offset-capable because the minimum radius is not adhered to. Many of the problems mentioned are due to the different data formats or due to interface problems and can add considerably longer time and harder work in the development process. This gives an example of the problems at the interface level between different disciplines.

Another important factor is the humans which run processes, e.g.: In the ideal scientific world, all scientists from different disciplines should work together to solve problems. The reality, however, is very similar to the world of industry. The pressure to publish something innovative, intellectual property rights and personal vanity are obstacles which prevent access to state-of-the-art know-how and lead to a transfiguration and academic use of material that is, at least in parts, already of general knowledge (Lakhani et al, 2007).

One of the most valuable commodities in countries that are poor in raw materials is the know-how possessed by scientists and industrial experts. A policy of non disclosure thus impairs today the integration of knowledge in parametric associative design models in the industry, as well as the continuous introduction of integrated processes that could penetrate beyond departmental divisions, business areas and company gates.

By means of a systematic design of CAD models, the parametric associative design allows the coordinated combination of interacting in one hand with structures that are packed with know-how and on the other hand with isolated partial structures containing little know-how. Often it is not the stored design procedure which is confidential, but rather the geometrical data and parameters of a new vehicle which result from this procedure.

A neutralisation of the CAD models by means of an exchange of confidential references versus standard geometries and standard parameters and precise agreements for usage could contribute to the application and optimisation of the models through a large number of users.

A possibility for extending the use of knowledge could be CAD model structures which are created and expanded on a common basis, whereby, the concept developer would contribute the geometry and the required function, and the production planner would provide the demands for the production of the designed system.

An intermediate goal for companies should be the establishment of libraries containing PAD models with standardised “Best Practice” solutions. A prerequisite for this would be a corporate culture which encourages close cooperation and provides its employees with a comprehensive understanding of relationships in processes, so that they can put their expertise at the disposal of a third party.

This leads to the research question:

> ***"How does the parametric associative design support the development of body assemblies with distributed tasks?"***

And from the application point of view, the even more specific question:

> ***"How could a parametric associative CAD model be clearly structured to provide information for linked processes in the form of Knowledge Based Engineering (KBE)?"***

1.3 Aims and Objectives

The overall aim of this investigation is to determine how CAD models of typical automotive body assemblies could be defined to allow a continuous optimisation of the number of iterations required for the final design and the number of variants on the basis of automated update-safe parametric associative design (PAD) models.

The first objective would be to produce computer models as test benches and demonstrators of the PAD applied to automotive body engineering.

The original contribution of this research is the application of PAD to the process and design methods of automotive body engineering. The novelty in this approach and contribution to knowledge is the application of PAD in the process chain of automotive body development where the combination of functions and forms of assemblies often is more complex than in mechanical engineering.

In actual processes a lot of knowledge gets lost from project to project for several reasons, e.g. change of responsibility, new project team etc. The second objective of this thesis is to define structures where the explicit knowledge of the company and implicit knowledge of the engineers involved in the project, can be captured and transported between various process steps to improve the time and the quality of the output. Automotive body design methods of are the main focus of this investigation. Rules and standards need to be defined to allow consistent update safe PAD models which can be re-used in follow up processes and other projects.

The equal distribution of design work between design engineers is another aim of the work. Large assemblies can either be subdivided according to the bill of materials or subdivided into zones. For the third and fourth objective the zone based approach needs to be investigated to find suitable rules of how far the degree of detailing of defined zones is reasonable and how a pure section based concept development can be substituted by reusable models of process chains from concept section to 3D assembly zones.

1.4 Methodology

In 2001 the author started to explore the field of PAD in depth. Two PAD training courses in 1999 and 2001 and 3 month industry training at an OEM in 2001 were the basis of this investigation. In 2000 the author started lecturing Parametric Design at Hamburg University of Applied Sciences.

From 2003 the author developed a new study course named, “The Design of Body Assemblies with Distributed Tasks using the Help of Parametric Associative Design” at the Hamburg University of Applied Sciences to research methods of parametric design and cooperative work under supervision of the author with groups of students, engineers from industry and university. 25 tasks for group work were defined by the automotive industry and investigated and designed by 25 groups of five to eight students in a period of 7 months. In a number of about 70 further joint projects (final year projects of 5 months duration) supervised by the author between university and industry the focal points and methods for product development and links have been investigated. In the centre of the new group work seminar the following subjects are of main importance:

- to find, apply and improve design methods for automotive body design,

- to distribute the planning and design work in an equal manner,
- to test Knowledge Based Engineering (KBE) applications,
- to reproduce the process chain of product development in the PAD models.

Examples of the 25 projects are closures such as bonnet, side doors and tail gates, front end, floor panel or rear wall of Body In White (BIW) or building sets for motor cycles or heating, ventilation and air conditioning (HVAC) system. The results of the projects have been presented to guests from the industry and universities in four conferences organised by the author.

The investigation for this present report is organised according to three different foci / working steps:

Step 1 of the investigation deals with the exploration of the PAD technology and the implementation of the complex new CAD-methods into the development process of the OEMs and the suppliers integrated in such processes. The complexity requires a step-by-step consideration of advanced detail parts designed with driving parameters on a part level only up to relational design models with links throughout the process chain (see levels of implication in chapter 3). Standards have to be defined, such as standard structural trees, data exchange formats etc. as well as standardised design steps for typical body assemblies. The permanent outsourcing of design work throughout the last two decades has led to a loss of design knowledge on the part of the OEMs today and prevents the OEMs from taking full advantage of the new methods.

Step 2 of the investigation deals with the product development process. The idea is to represent the development processes in PAD models, where the standard processes and the vehicle processes have to be explored. The vehicle process in body engineering can be divided into two main phases, the preparatory or concept phase and the serial (mass production) or detailing phase. The concept phase is a phase of strong collaboration between styling, package, concept design, suppliers and digital prototyping. Results from calculations and simulations provide strategies to optimise the parameters defined between simulation and design. This phase with a wide span of design variants and design changes is predestinated for the uses of

PAD while the content of parametric and association should be reduced in the detailing phase as soon as design decisions have been made. In both phases a lot of changes happen and the developments are optimised in several loops.

Step 3 of this investigation concentrates on specific items of PAD to offer a deep insight into important study areas. The idea is to explore design methods for surface design on one hand and the distribution of work between groups of engineers working in concept phase on the other hand. Collateral CAD-validation test beds with different levels of complexity prove the theories defined.

1.5 Conclusion to the Chapter

This chapter describes the situation in the automotive industry and the role of the body engineer. A comparison of old and new design methods shows the chances of a new product development under strong support of parametric associative design. This leads to the research question and an explanation of aims, objectives and methodology.

The next chapters will point out important differences between body engineering and mechanical engineering and give an insight into the fundamentals of parametric associative design.

Comments:

The following considerations are based largely on generally accepted formulations. However, it should be pointed out that the results based on body design analyses obtained using CATIA of Dassault-Systèmes. It is therefore possible that some quotations are latently subject to an interpretation that depends on this specific system.

In the application of CAD systems on an international level, the use of English menus has prevailed. In the terminology which refers to specific systems, these were replaced by neutral expressions wherever possible in this study. Complex terms are further explained in the Glossary and in the List of Abbreviation.

Chapter 2 Automotive Body Design Methods

This chapter explains basic and advanced methods of automotive body design which are required for the definition of stable parametric associative design models / processes and for the validation test bed.

2.1 Classification of Surfaces and Surface Continuity

The interior and exterior aesthetic of automobiles (see figure 2.1) strongly depends on the quality of the exterior and interior surfaces (Class A) defined by visible light and shadow flows and highlights and the assembly of the body panels and functional components (e.g. windows and lights), i.e. the visible gaps and the relative alignment of the sub-assemblies. Flushness of surface transitions and the optical appearance of gaps define the quality of an automotive body surface. To give an example,

Fig. 2.1 Quality of gap design in the example of an AUDI (Dehn, 2001)

gaps have a functional relevance e.g. to split movable parts from fixed parts of the body as well as an aesthetic relevance to draw characteristic styling lines. To keep the optical appearance of a gap as small and as parallel as possible tolerances and thermal expansions of the parts involved have to be taken into account and the gap should run always parallel to the theoretical radius run-out of a surface. To keep the optical appearance of a sloped gap parallel the designer will change the technical breadth of the gap and probably displace the flush mating surfaces.

The method of designing visible interior and exterior surfaces (Class A surfaces) differs among auto manufacturers. In most of cases, point cloud and mesh data provide the basis for modelling the surfaces and gaps according to technical and aesthetical requirements. A common tool for the bridging between styling and engineering is the technical surfacing software ICEMSurf (Lender, 2001).

The technically modelled surfaces are the basis for the engineering design of the car body panels. Class B surfaces (grey zone/hidden zone surfaces) are surfaces which become visible when the side door or the trunk lid is opened. They are initially generated by the concept engineers as functional surfaces when designing doors and body-in-white panels and subsequently remodelled by Class A engineering to meet the aesthetic requirements

The following figure shows the classification of automotive body surfaces as defined by BMW:

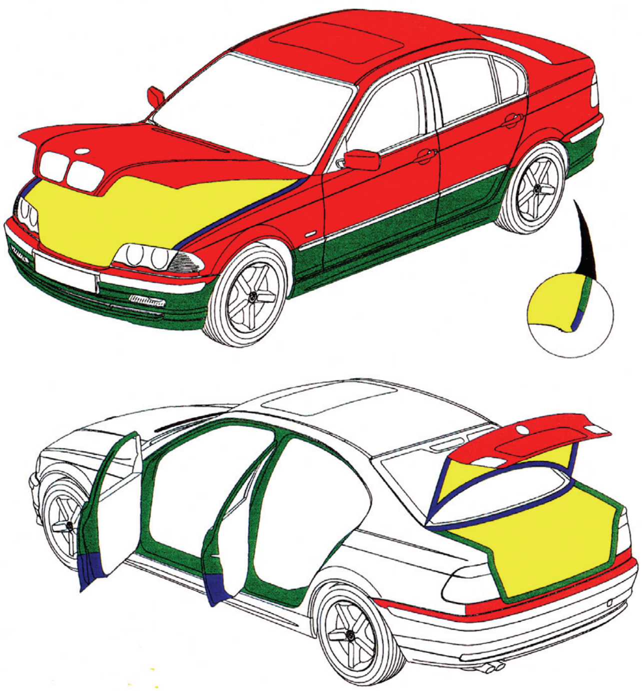


Fig. 2.2 Classification of surfaces according to BMW Class A, Class B, Class C, Class D (Dehn, 2001)

Visible surfaces should ideally have a continuous curvature gradient (G3); in any case they must have curvature continuity (G2) (see figures 2.3, 2.4 and glossary).

To visualize the continuity of surfaces or surface transitions, CAD software displays a porcupine-type (comb-type) curve along the cross-sections of the surfaces or curves. The principle of these porcupine curves is that the curvature radius is visualized by a straight line (spike) normal to the analysed curve, pointing away from the centre of the curvature. The length of each straight line is proportional to the curvature radius. The curvature gradient (comb-curve) is visualised by a smooth indicator curve connecting the end points of the straight lines (see figures below).

The design of car body panels on the basis of design surfaces requires faces, surfaces, and curves with unambiguous directions.

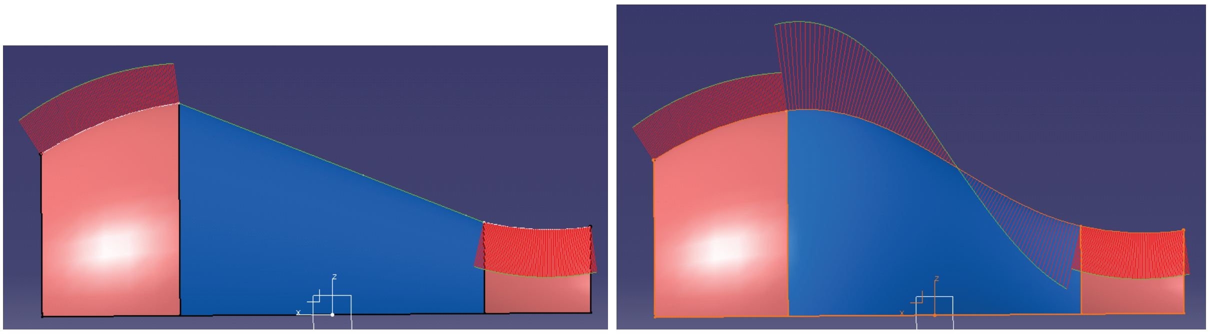


Fig. 2.3 Porcupine-type curve analysis, Point continuity (G0) (LHS),
Porcupine-type curve analysis, Tangent continuity (G1) (RHS)

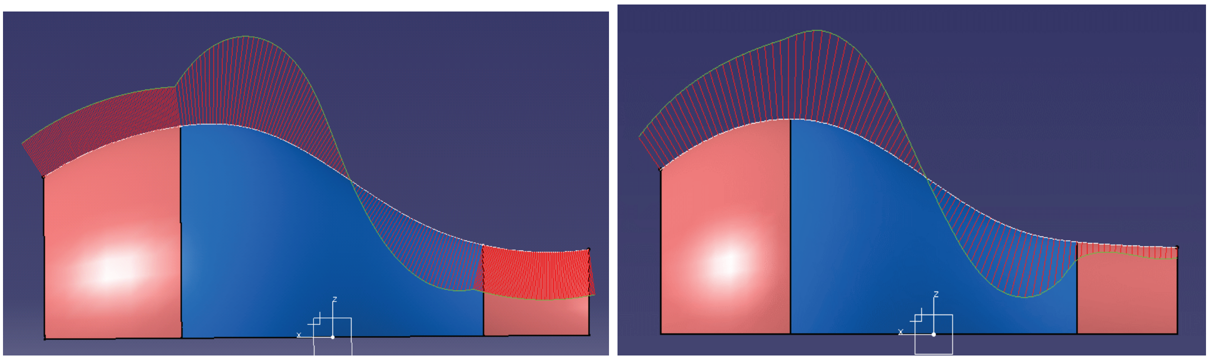


Fig. 2.4 Porcupine-type curve analysis, Curvature continuity (G2) (LHS),
Porcupine-type curve analysis, higher continuities (G3/G4) (RHS)

2.2 Direction Control for Curves and Surfaces

Contrary to general mechanical engineering, where individual parts are often designed with individual, independent coordinate systems reflecting their orientation during manufacturing processes, car-specific body parts are designed with reference to the vehicle coordinates system, oriented and located in the assembled car-line position. Only approximately 10% (Braß, 2004) of the parts of an automotive body-in-white are carry-over or standard parts with their own individual coordinate system. During the initial phase of package design and during the phase of concept development (simultaneous with styling), the final location of the sub-assemblies is not yet defined. In these early phases of product definition, sub-assemblies of the automotive body may be moved with the aid of local coordinate systems.

Car body panels are designed in their car-line position. The panels are curved and inclined once or twice and their orientations are neither normal nor parallel to the main planes of the vehicle coordinate system.

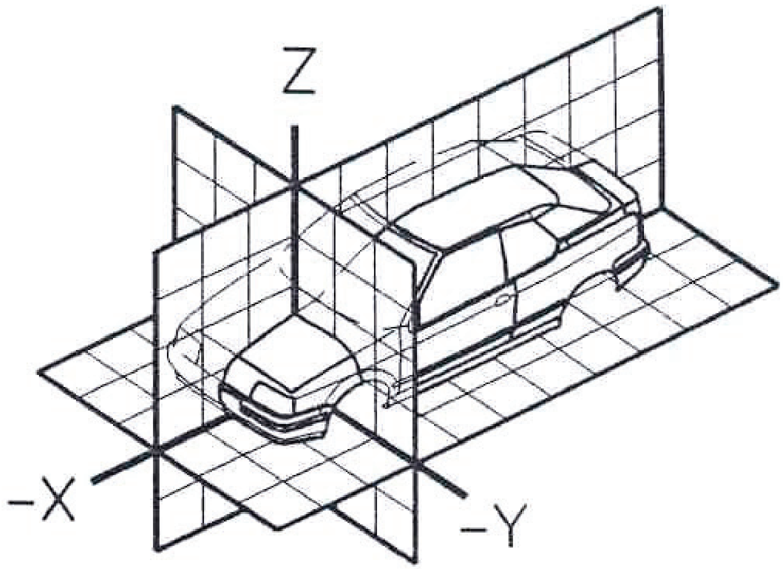


Fig. 2.5 Vehicle coordinates system (Karmann, 2005)

Generally, only the left half of the car is designed, while the right half is generated as a mirror-image of the left half. Most of car manufacturers place the origin of their main coordinate system in the centre of the front axle (see figure 2.5). The location of each body part in the car is clearly defined with reference to this vehicle coordinates system, using the following three coordinates: X (distance from the centre of the front axle in the direction of the car length. Only Ford defines the vehicle coordinate system in front of the car in a certain distance from fire wall to avoid negative X-coordinates), Y (distance from the centre of the car in the direction of the car width), and Z (distance from the centre of the front axle in the direction of the car height).
As many body parts are made of panels with constant sheet thickness, shape design is the standard method for designing body parts. This means that either the interior or

the exterior surface of the part is designed. Subsequently, a solid model is generated by applying a material thickness to the shape model. The solid model is then used to determine mechanical properties like weight etc., and for Digital Mock-Up (DMU) processes.

The design of these surface models requires the unambiguous definition of the surface and curve orientation. This is achieved with the following procedure:

The orientation of curves is defined strictly according to the vehicle coordinate system. This means that horizontal curves which are normal to the front axle (i.e. in the direction of the length of the car) are defined as parallel to the X axis. Horizontal curves which are parallel to the front axle (i.e. in the direction of the width of the car) are defined as parallel to the Y axis. Vertical curves which are normal to the front axle are defined as parallel to the Z axis of the vehicle coordinate system. Closed curves are defined in the mathematical positive direction, depending on the main view rendering an image of their geometry which comes closest to its true image.

Parts with visible surfaces (Class A and Class B) are always designed starting off with the modelling of the visible surface. All other parts are designed starting off with the punch (die) side of the part (see figure 2.6). Parts with varying thickness (e.g. a cast part with ribs) are designed by defining all outside and inside surfaces, with the surface normal pointing inwards.

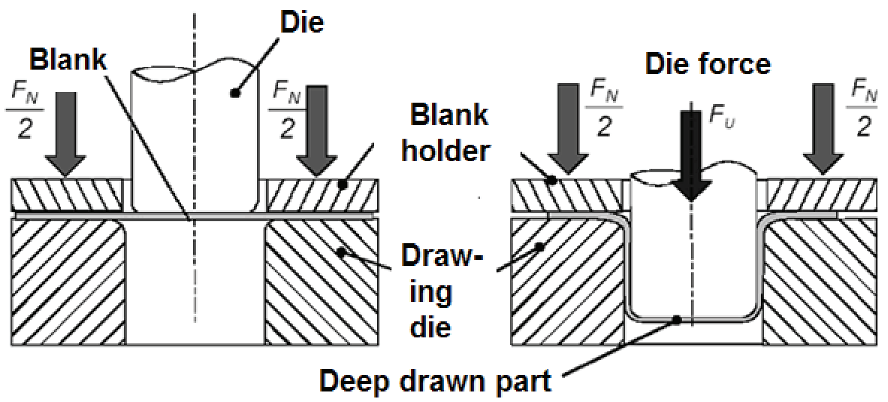


Fig. 2.6 Principal design of a drawing tool (Schmidt-Jürgensen, 2002)

In CAD models, the designed surface must be marked unambiguously. This can be done by ensuring that the surface normal and the sheet thickness vector always point in the direction of the sheet thickness (see figure 2.7).

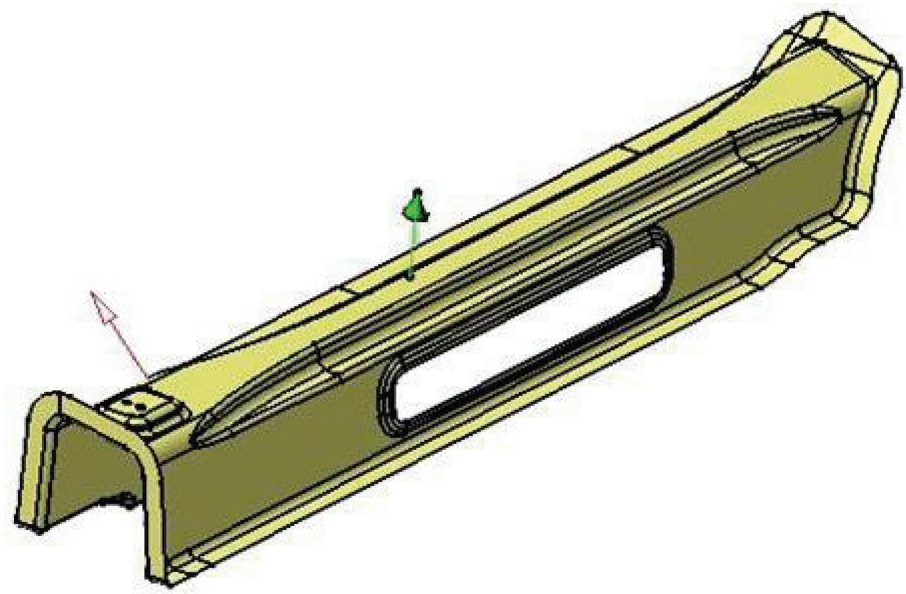


Fig. 2.7 Cross member seat (punch surface) with surface normal (mauve) and sheet thickness vector (green; length of arrow = sheet thickness x 100) (Rehner & Brill, 2006)

2.3 CAD Techniques for the Design of Body Parts

This section describes the different CAD techniques used to design body parts according to the different needs defined by design process and manufacturing methods.

2.3.1 Hybrid Design

The complex compound angle of surfaced parts of the automotive body and its interior are usually modelled as three-dimensional hybrid designs. Hybrid design is a mixture of wire frame, surface and solid design. Points and curves define the basic geometry of the surface models. Surface models are used to describe the part geometry as well as the cutting geometry (see figure 2.8).

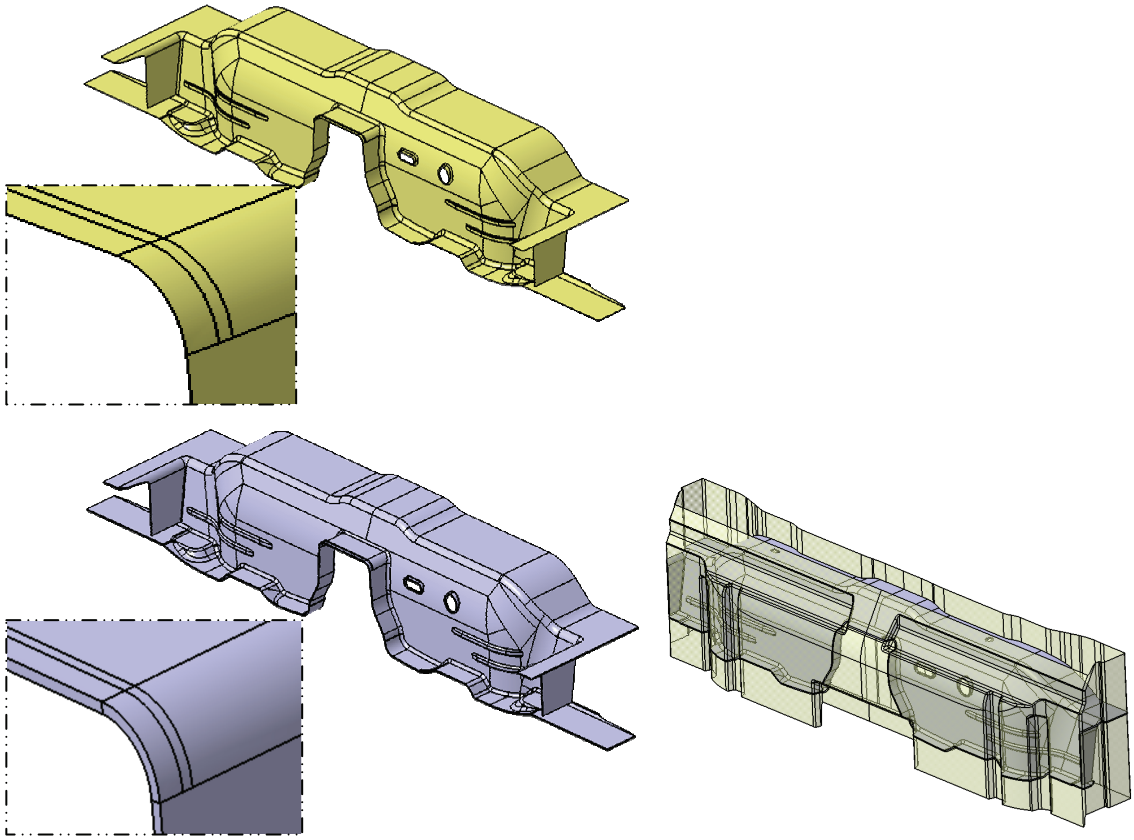


Fig. 2.8 The surface model of the sheet metal part (yellow) is thickened in the direction of the sheet thickness vector to become a solid model (grey) and be trimmed by the cutting surfaces model (beige) (Brill, 2006)

The accomplished surface models of the parts are thickened to obtain solid models. Typically, ribs, domes and draft angles are added to the solid model to accomplish the design. Additional information contained in the solid models (material, weight, centre of gravity etc.) increases the usefulness of the CAD models.

Modern CAD software facilitates the structuring of hybrid designs and the interaction between the workbenches of the surface and solid modelling. To ensure that the models can be updated in a stable manner (without the model “crashing out”), the Boundary Representing elements (BReps), e.g. faces of solid elements may not be used for surface modelling.

2.3.2 Visualisation of 3D Models & Wire Frame Representation

A wire frame model is the basic model for defining and visualizing 3D designs. In 2D design of automotive bodies, the wire frame model representation is the only possible method of visualization. The free-form automotive surfaces are represented by three-dimensional curves (shape-defining guide curves = character lines) and planar curves (grid-sections = height-, breadth- and length-sections). For perspective visualisations, boundary contours are added to obtain an optimum three-dimensional visualisation of the surfaces by a 3D mesh of wires (see figure 2.9).

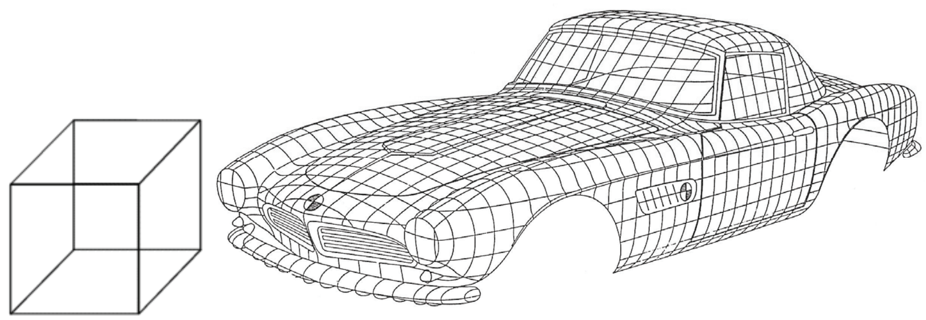


Fig. 2.9 Wire frame model (edge model) of a cube and wire frame model of an automotive body (BMW 507 – Schäpe, BMW)

Wire frame models form the basis of three-dimensional surface modelling. The elements of the wire frame model are points, straight lines and curves. Straight lines and curves are calculated from several supporting points using interpolation or approximation algorithms (see figure 2.10). The calculated contours are used for the generative surface modelling.

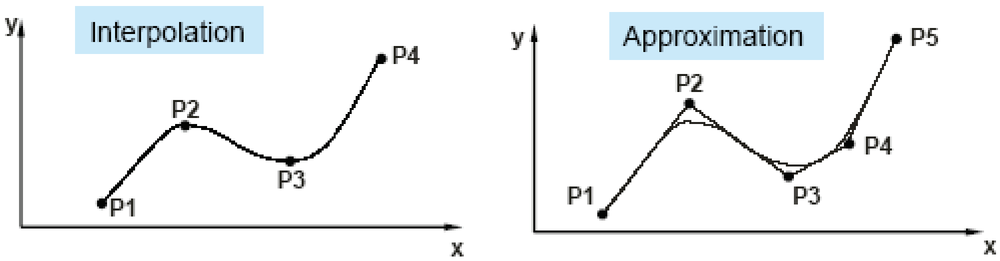


Fig. 2.10 Interpolation and approximation of curves from points (Lindemann, 2005)

The design of cross-sections is a well-approved method in automotive body design to analyse and define package zones. Together with guide curves and reference surfaces, these cross-sections are the basis of the surface modelling process.
Points define the basis of guide curves. Specific reference points (points of interest) used for positioning nodes or for controlling/manipulating profile contours are defined on the spine curves. The profile contours located on the normal planes and the spine curves are the generative input to the surface design process (see figure 2.11).

2.3.3 Surface Creation & Definition

The basic curved surface types used in the surface modelling process are profile surfaces, skewed surfaces and free-form surfaces.

To improve their quality, surfaces should always be designed using planar curves with at least continuous curvature. Although modern CAD software permits the construction of surfaces from three-dimensional curves, however, these methods should be avoided wherever possible because such surfaces are difficult to control.

To design a profile surface, a planar closed or open profile contour, a guide curve and a spine are required. The profile contour can consist of one or several straight lines and/or curves. The spine curve must have continuous curvature. Planar spine curves with continuous curvatures yield the best surfacing results. The guide curve must have at least tangent continuity.

The size (i.e. length and height) of the profile contour and the curvature of the spine curve are directly interrelated. The combination of a guide curve with small curvature radius with a large profile contour leads to undesirable loops in the derived surfaces.

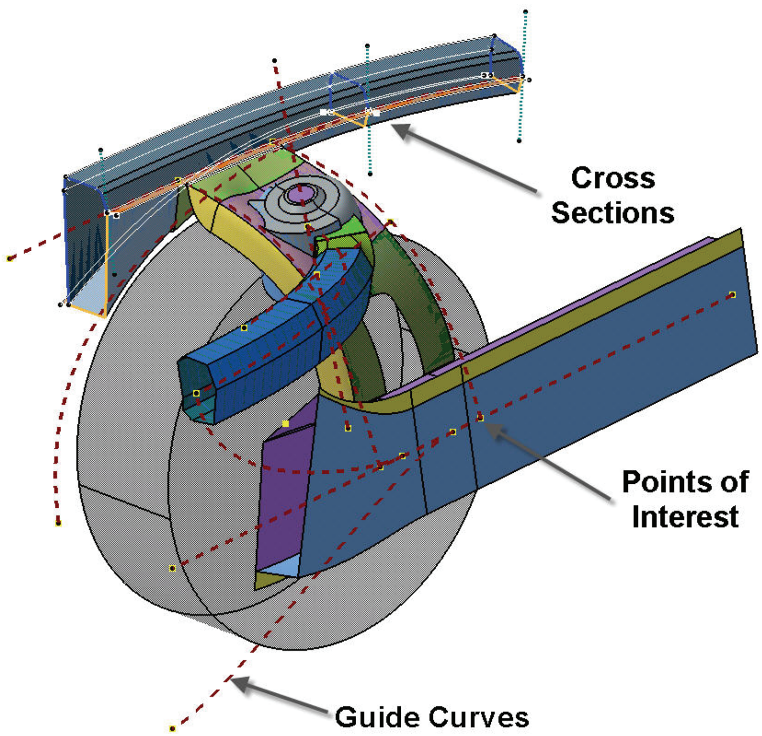


Fig. 2.11 Design of a body structure (front end) on the basis of a wire frame model (Rösen, 2006)

The following figures show the profile surface variants. Figure 2.12a shows a prismatic profile surface, with spine and guide curve (violet) being identical. The profile is defined by the light blue coloured, planar contour. The non-prismatic profile surface in figure 2.12b has the same profile contour, and is defined by a separate spine curve (straight line = X axis) and a guide curve (violet):

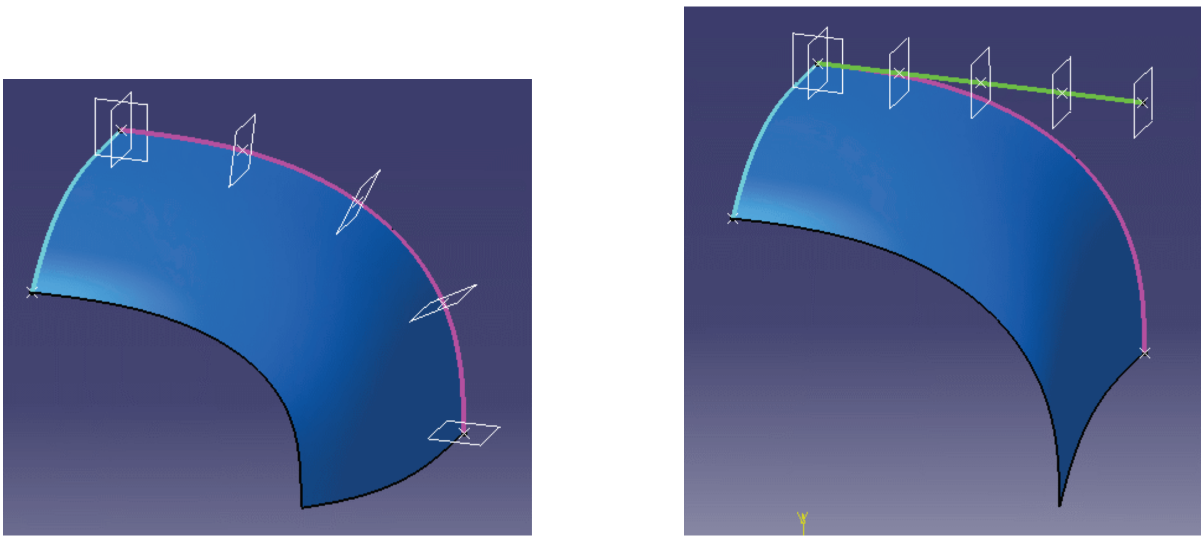


Fig. 2.12 Prismatic profile surface (swept surface) (LHS),
Common profile surface (swept surface) (RHS)

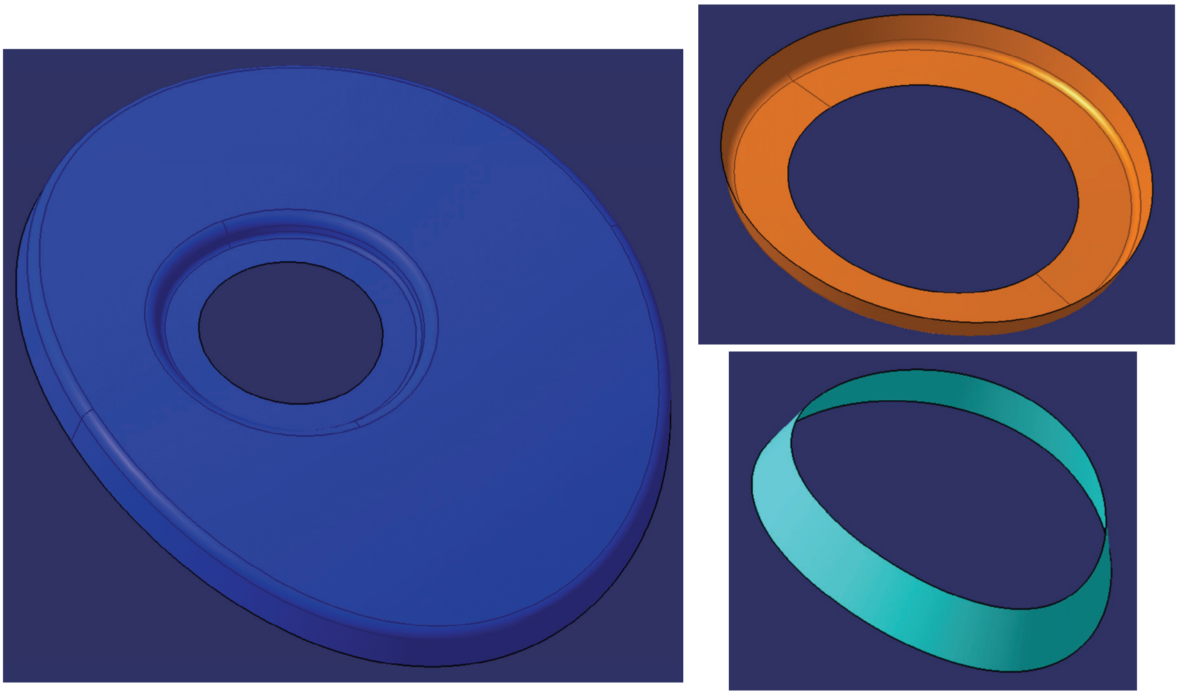


Fig. 2.13 Example of profile surfaces: A reinforcing body part on the basis of a free-form surface contains an outer stiffening collar flange (turquoise) on the basis of a common profile surface and a bedding flange (orange) for the mating part on the basis of a prismatic profile surface.

The boundaries of car body panels often have a prismatic cross-section (see figure 2.13). Examples of prismatic profile surfaces are the bedding surfaces for window panes, the hemming flanges on doors and closure panels or joining flanges (e.g. for spot welding).

General prismatic profile surfaces as shown in the right part of figure 2.12 are often applied to designs which must fulfil special requirements to ensure the parts can be removed from the mould/tool (e.g. central surface of seat upholstery or reinforcing collar of a body panel).

Skewed surfaces (see figure 2.14) are created with the aid of skewed straight lines. Skewed straight lines are neither parallel nor do they intersect each other. In CAD models, skewed surfaces are usually defined using two planar curves which are located on parallel planes.

Skewed surfaces e.g. are required when designing doors and closure panels, as transition surfaces in the corners of the inner panels.

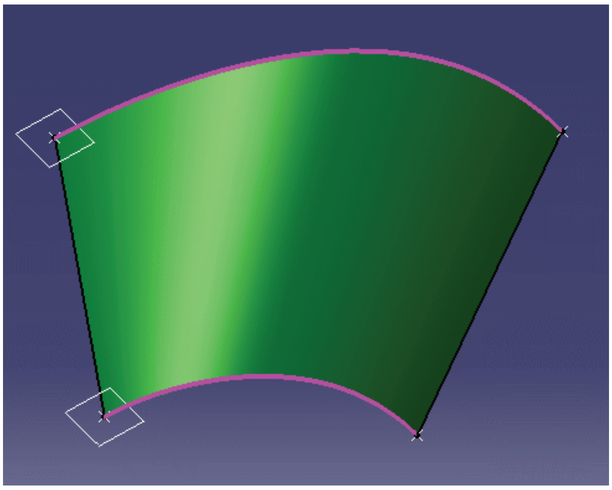


Fig. 2.14 Skewed surface

Together with profile surfaces, free-form surfaces are the most common type of surface used in automotive body design. Free-form surfaces are designed from four or at least three planar curves (see figure 2.15). One of these curves forms the theoretical boundary and acts as the spine curve. At least two other curves (cross-sections) are defined on planes normal to the spine curve.

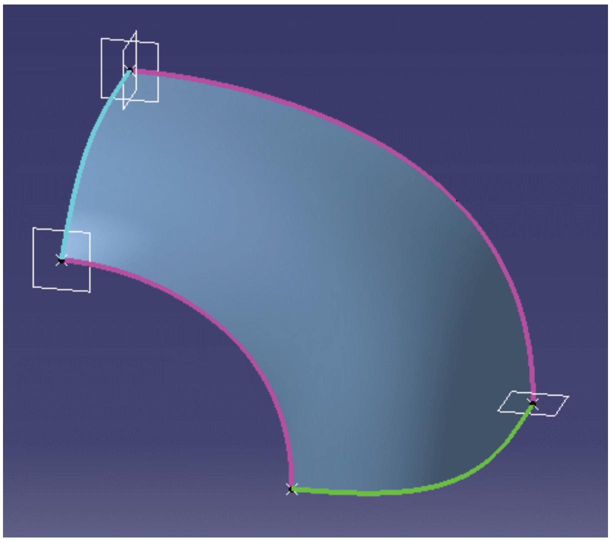


Fig. 2.15 Free-form surface

2.3.4 Solid Model Formation (Construction)

A solid model is normally composed of several solid bodies.

When solid models are made up from simple, basic bodies (primitives), they are referred to as Constructive Solid Geometry models (CSG – see figure 2.16). A solid geometry model made up from surfaces with defined material direction is called Boundary Representation elements (BRep) model.

The basic bodies (also referred to as primitives) of the CSG method have a shape which can be described with simple mathematics, e.g. cube, cylinder, prism, pyramid, sphere, torus or wedge with rectangular basic surface. Some software packages also permit the definition of CSG on curved surfaces. These basic bodies are then combined with Boolean operations like unite (U), subtract (–) or intersect (∩) to build the desired solid geometry models. The sequential data structure of CSG models offers a straightforward description of the design sequence and requires little memory space.

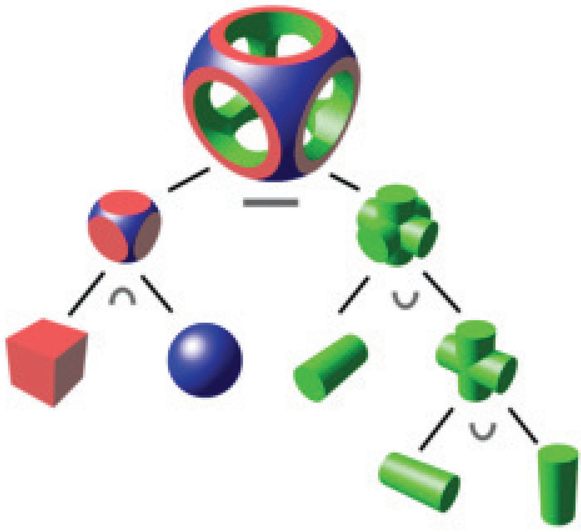


Fig. 2.16 Example of a CSG tree. Each node shows the symbol for the Boolean operation used (Wikipedia, 2007:1).

The BRep method defines solid bodies with the aid of the surrounding surfaces (skin) and their boundaries. These boundaries can consist of basic topological elements like nodes (corner points, vertices), edges, faces etc. Each of these basic elements has attributes which clearly describe its geometry (e.g. coordinates, curvature, angle etc.) and interrelations with neighbouring basic elements. Contrary to a pure surface model, where the designer is responsible for defining the direction of the surface normal, the data base of a BRep model contains material vectors which point towards the inside of the solid body it represents. For subsequent design steps (like trim, offset, fillet…), it is necessary to control these vectors and adjust them according to the required direction.

Generally, a solid basic body is designed by defining planes on which closed profiles are specified in sketches, representing the cross-section of the basic body. These sketches are then extruded in the direction normal to the plane on which they were defined, to achieve a three-dimensional body. Alternatively, basic bodies can be created by rotating an open profile around a rotation axis. A more complex method is the multi-sketch body which basically consists of several non-parallel sketches which are extruded and intersected with each other. These bodies are all referred to as sketch-based solids.

It is also possible to convert a surface model into a solid geometry model by sewing surfaces (provided they are fully closed) with virtual material or by adding a material thickness to the surfaces (on one or both sides of the surface).

2.4 Body Design Based on Sections and Functional Surfaces

Hänschke (2007) as well as Zimmer & Schumacher (2007) describe a process chain for body development with body engineers which define geometry only and Computer Aided Engineering (CAE) engineers which control the functions of a body. These are very single sided considerations because besides important functions like e.g. crash, safety, durability, Noise and Vibration Harshness (NVH) acoustics and aerodynamics there are hundreds of other important functions a body engineer has to look after, such as positioning of park distance sensors, air flow in a grill design, positioning of hinge axis and latch, layout design of a four link hinge to guarantee non interference between bonnet and front screen in a crash, side door movable glass fitting, kinematics of door stop etc.

When designing sub-assemblies of the automotive body one must distinguish between styling-driven and function-driven and purely functional (not visible for customer) sub-assemblies. A typical example of a sub-assembly driven by design plus function is the A-Pillar, the exterior surface of which is defined by the styling of the car. Functional surfaces for installing body components (like windows, doors, binocular obstruction of the driver) as well as secondary elements (like structural reinforcements, water hose, electric harness) and passive safety devices (like airbags) add functional geometry to the styling surfaces.

As an example of a merely function-driven sub-assembly defined by the author on basis of profiles, the design of a front seat cross member for a limousine shall be explained in this section.

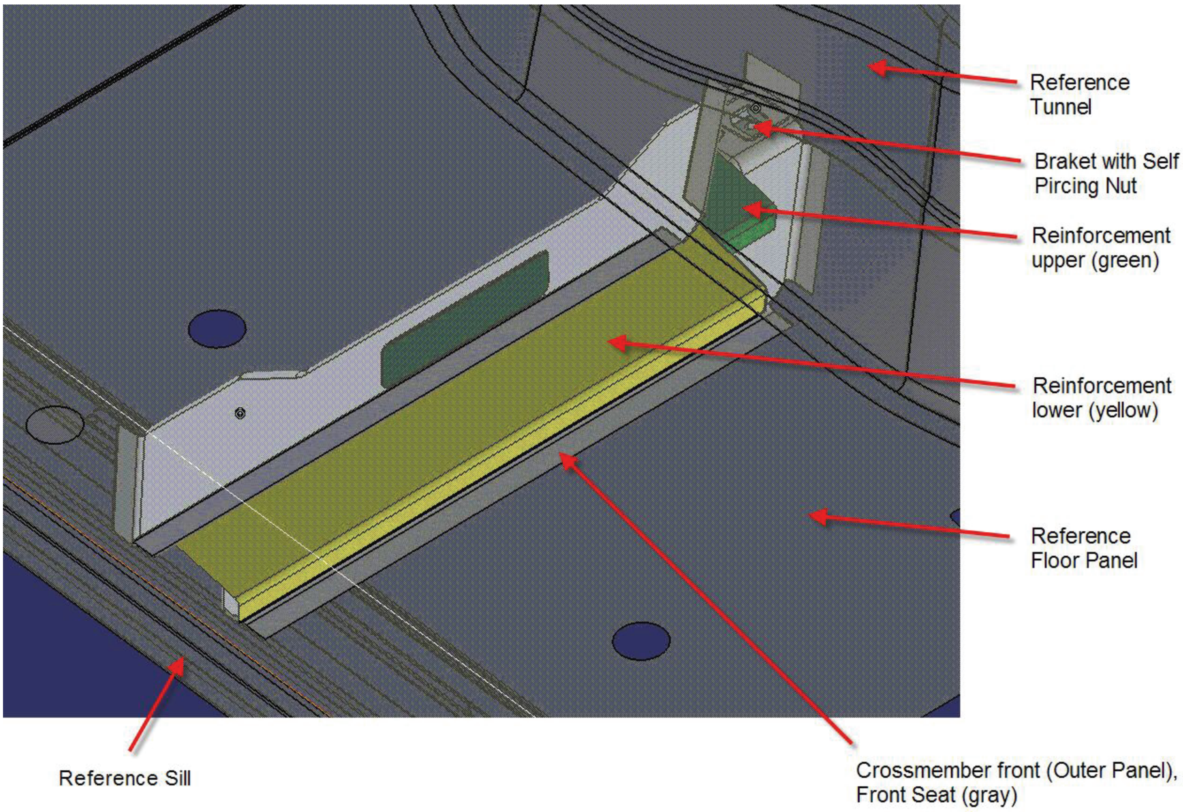


Fig. 2.17 Assembly cross member front seat in context with mating BIW parts

The design of the sheet metal parts of a cross member assembly is typical for simple profile driven parts. The bracket with nut for seat mounting is taken as an example for Carry Over Parts (COP). The reuse of the corner reinforcing beads geometry gives an example for the use of templates. The PAD functions and the design in context link methods therefore will be explained in depth in a later chapter.

The seat cross member is a structural element of the automotive body-in-white (BIW) which serves two functions: adjustability of the seat position, and secure sustainment of all loads, particularly in the case of an accident.

The analysed seat cross member is designed for a car with an aluminium body (AUDI A8), and it shall be manufactured from sheet metal (i.e. from several panels which are joined together). The whole cross member front seat consists of an outer panel, two horizontal reinforcement panels and two brackets with self piercing nuts for attaching the seat (see figure 2.17). The reinforcement panels, the support panels and the outer panel are joined together with self piercing rivets (not shown).

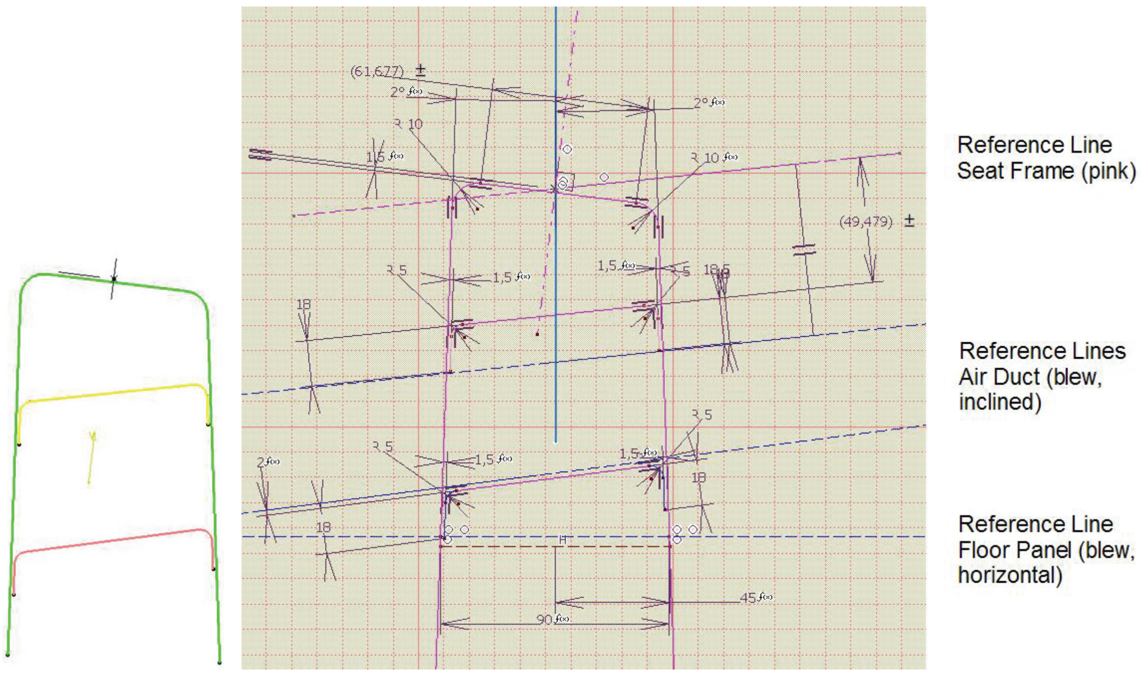


Fig. 2.18 Master section for developing primary prismatic profile surfaces

The design is performed as Relational Design in the context of all neighbouring parts. The bottom of the seat cross member must mate with the floor panels. The sides of the seat cross member are limited by the position and geometry of the sill (outer side) and the tunnel (inner side). The upper surface is determined by the geometry and the attachment of the seat frame. The ventilation duct connecting the air conditioning module (HVAC) with the rear foot compartment crosses the cross member front seat. Control devices shall be mounted to the floor panels in the area of the cross members. The joints between the outer panel of the cross member and the sill, the floor panels, and the tunnel shall be designed as butt joints (with bent portion flanges), which are MIG welded. The geometry must be designed to ensure rapid moistening and draining of the surfaces in the process steps of immersion painting (cataphoretic painting). All geometric reference elements of the neighbouring parts are prepared in separate adapter models according to the Input-Design-Output (IDO) principle (see chapter 3) and supplied to the design of the cross member.

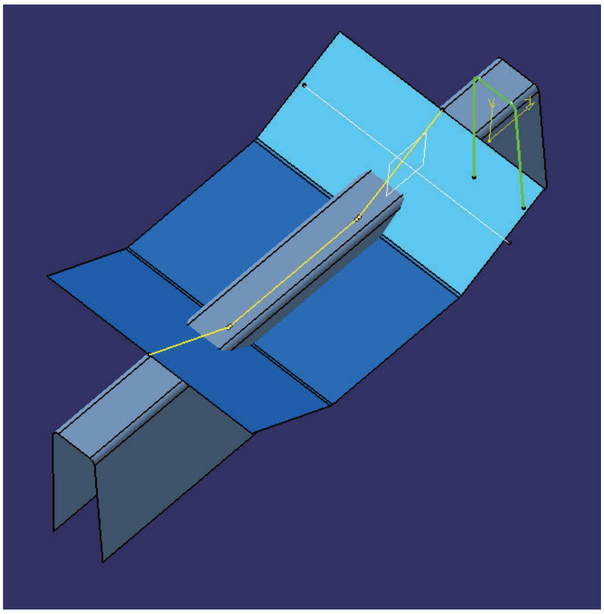


Fig. 2.19 Reference geometries and primary surface of the outer shell narrowed down by a second primary surface to give space for the seating pan.

The design is performed on the basis of a true development (compound radial) section, where the outer panel and the horizontal reinforcement panels are dimensioned relative to the reference geometry (see figure 2.18). The location of the cross-section geometry is also determined by the reference geometry. The cross-sections are used to extrude primary surfaces with planar prismatic profile. In the centre, the outer panel is indented by a second surface with prismatic profile, to account for the space reserved for the seating pan (see figure 2.19). The design of the seat attachment depends on the design of the seat frame. The flanges and the aperture for the ventilation duct are designed according to manufacturing aspects in context with the corresponding parts. To reinforce the outer panel, its design is accomplished with four “bird’s beaks” (corner reinforcing beads) (see figure 2.20).

The replacement or the modification of reference geometry causes the automatic adaptation of the parts modelled with the parametric associative design method, to reflect the design intent implemented in the model.

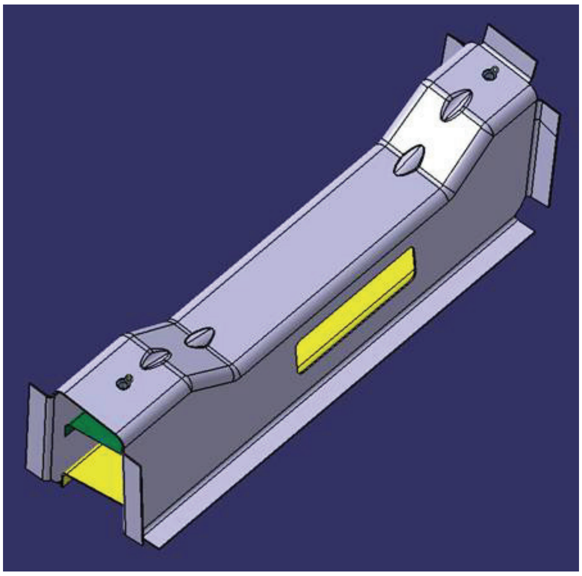


Fig. 2.20 Completed Assembly Cross member front Front Seat

2.5 Conclusion

The methods and examples shown in this chapter point out important basic features of body design which are the preparation for a structured parametric associative design. In the following chapters the fundamentals of PAD will be explained and in this context the methods of body design will be articulated further.

Chapter 3. Fundamentals of Parametric Associative Design - Consolidated Findings

Until the mid 80's, Class A designs, design concepts and detail drawings were prepared manually on drawing tables and drawing boards. Car body designs were drawn in the scale of 1:1, requiring drawing tables 7 metres long and 1.5 metres wide (see figure 3.1).

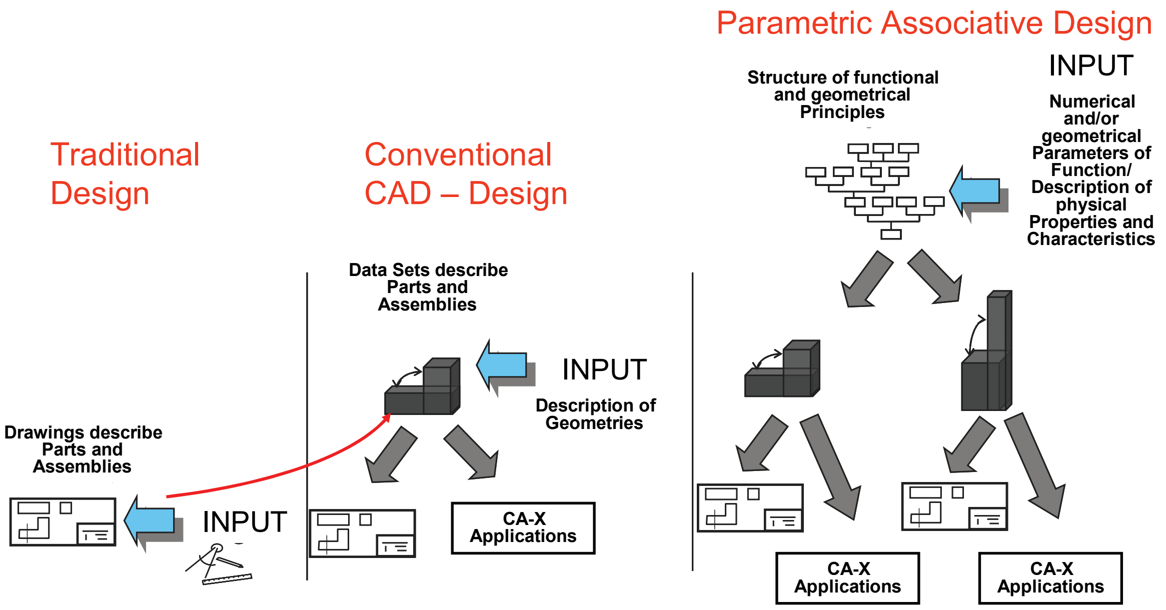


Fig. 3.1 Comparison of design methods (Albers, 2001)

The designs were prepared manually on non warping drawing film with special pencils, felt tip and ink pens, using set squares, curve templates, spines with weights, and compasses. The required quality of the drawings with respect to line thickness, clearness and geometric correctness was extremely high as developed 2D curves (form lines) and 3D curves (character curves, functional curves) of the drawings were directly used for car body manufacturing.

The design process was always focused on complete assemblies and package areas. In most cases, several designers prepared their drawings in a team, working simultaneously around the table, discussing problems and necessary adjustments or corrections as they arose. Communication was not a problem.

As early as 1975, car manufacturers started experimenting with computer-aided design software. Initially, two-dimensional drawings were created on the computer screen, and the three-dimensional models were later derived from the drawings (see figure 3.1). However, car body designers soon realised that the design process had to be inverted, starting off with the modelling of the 3D geometry and subsequently deriving the drawings. OEM's and independent software companies developed a large variety of CAD programs, which were not as comprehensive as today's software, but contained several very useful features. According to the experiences of the author examples of such pioneering software solutions are Ford's "PDGS" and Daimler's "SYRKO".

Design with non-parametric non-associative CAD software is focused on the modelling of individual parts and components. Complete assemblies are difficult to design with these systems. To check how the individual parts fit together in an assembly, session and digital mock-up models were generated. The designers started to concentrate on their individual parts, finding it difficult to keep the requirements and restrictions of the final product in mind. Communication among designers working on separate computer work stations, often in different rooms or even at different locations, became increasingly difficult.

To overcome these problems, automotive manufacturers started experimenting with parametric associative CAD software in car body design projects in the mid 1990's (see figure 3.1). They followed the foot prints of power train developers, who had introduced the method of PAD about 5 to 10 years earlier, encountering fewer difficulties as their mechanical designs and solid geometry models were much easier to control with parametric and associative relations. The associative approach of PAD software used in automotive body design is focused on modelling assemblies as a whole and using reference elements and surface geometry to describe the boundaries of package areas.

A drawback of conventional, non-parametric CAD software is the fact that later changes often require complete remodelling of the geometry. The approach of PAD systems permits planned changes and variant generation through parameter modification or replacement of geometric objects. Thus, 3D models can be reused. However, before such "long-life" models are created, the designer must develop a strategy for efficiently and safely controlling his geometry through intelligent parameters and well-defined interfaces between components or package areas. Professional modelling methods must be developed and tested before being used in the design process to ensure stable, high quality design results. The desired output

of the PAD process is a reusable 3D model of the product geometry, containing related information for subsequent process steps like simulation, production planning, NC program generation, etc.
The objective of the PAD approach is to perform all steps of the product development process with the aid of 3D master models using native data from one CAD system. Additionally, all information pertaining to these process steps shall be linked to the models or incorporated into them. Thus, PAD models assume the role of key information carriers through the entire process chain from package design to manufacturing, and even for subsequent steps in the product lifecycle. With increasing integration of confidential information and knowledge into these models, new procedures are required for collaboration between automobile manufacturers and their subcontractors.

3.1 What are the main Attributes of PAD?

Apart from geometric information, PAD models contain the design intent in the form of dependencies between geometric elements. When certain design characteristics (parameters) are modified or replaced, the geometric model is automatically updated to reflect the changes. The update process relies on the design intent stored in the form of relations and associative links.

Usually, sheet metal parts are designed from several independent basic geometric objects like planes, points and curves, using a sequence of structured design steps. The chronology of the design steps and the parameters of the basic geometric objects are saved in a 3D model. Parameters can be of numeric (variables) or geometric (curves, surfaces etc) nature. They are defined through relations and controlled by rules and checks.

PAD offers optimum possibilities for documenting design intent, rapid modification of models to changing geometric requirements, the re-use of design steps ranging from small sets of elements and operations (micro-templates) to large product models (macro templates) as well as the rapid generation of part families from one CAD model.

PAD permits the definition of links and relations between parts and assemblies to control the transfer of information to dependent CAD models used in subsequent process steps or to trigger subsequent changes.

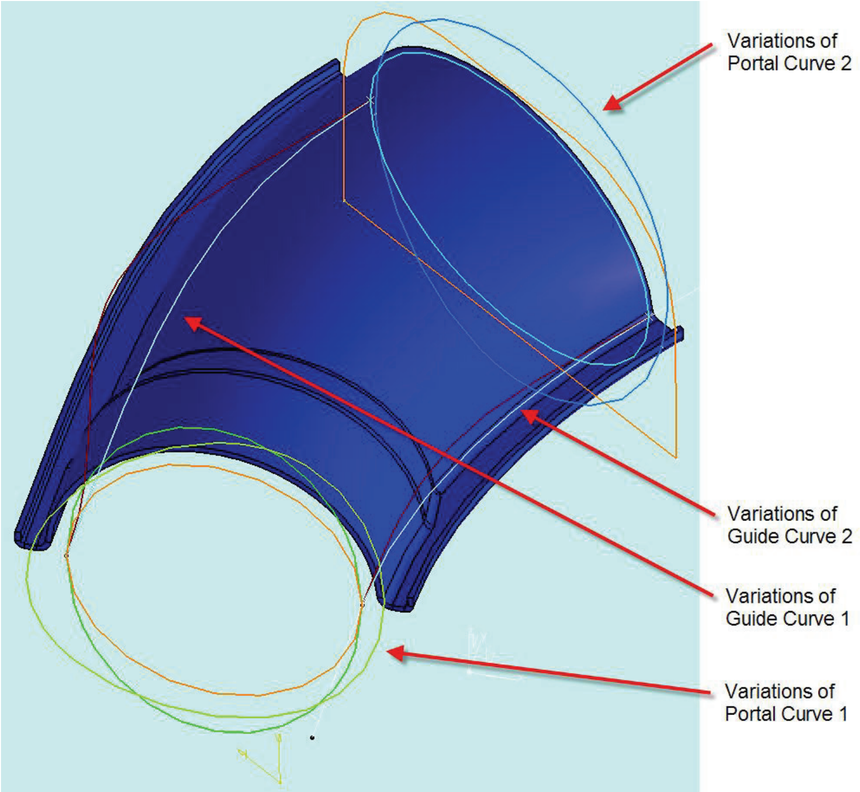
Compared with non-parametric CAD models, the file size of PAD models is larger because the complete design history is saved in the model.

To ensure better understanding of the PAD models, standard modelling methods must be used. However, the clarity of PAD models decreases with growing number of basic geometric elements used for the design because the number of relations rises. Such complex models become difficult to edit and may cause undesired effects when updating the changed model.

Consequently, the process chain of parametric associative CAD design requires a consistent structured approach with defined interfaces to a larger number of small, comprehensible CAD models.

3.2 The logical Operators of Parametric and Knowledge Based Engineering (KBE)

Modern CAD-Software such as CATIA V5, NX or ProE can store the explicit knowledge of a company and the implicit knowledge of the designer and reuse it for Design (see Glossary “Knowledge ware”). In the following sections the operators for Knowledge Based Engineering (KBE) are described.



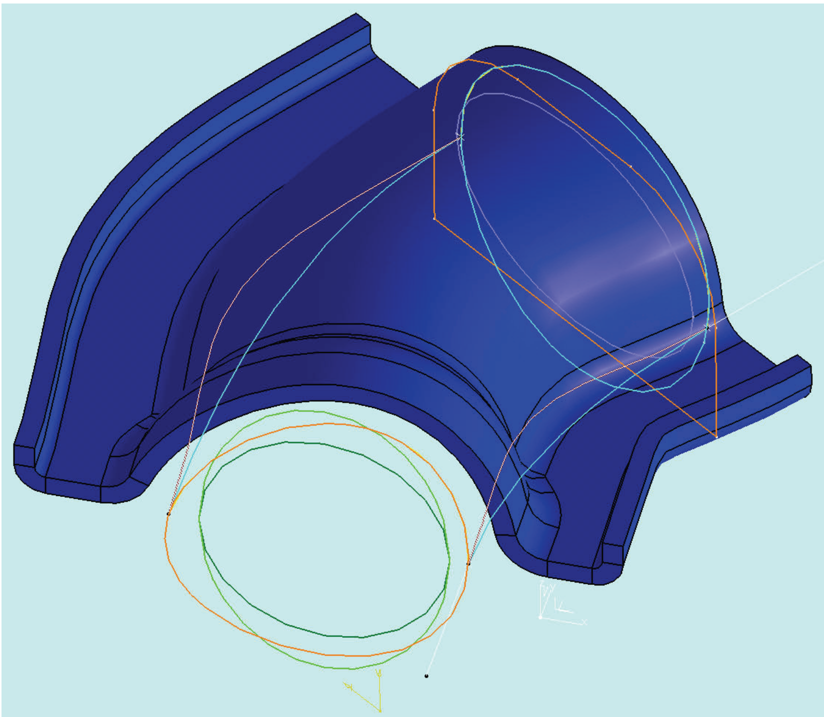


Fig. 3.2 Examples of design variants of a housing upper shell:
The numeric parameters for Sheet Thickness and Offset Width, and the geometric parameters Portal Curves and Guide Curves permit the generation of 120 variants (multiplication of all parameters defined).

3.2.1 Parameters

Parameters are control variables or elements used to define the geometrical form and the functionality of parts and assemblies. The modifications of these control variables or the replacement of elements causes the automatic adjustment of the modelled geometry according to the design intent stored in the PAD model. Parameters of the types “length” and “angle” are used to control the designed geometry. Modern parametric CAD systems additionally offer various different parameter types based on units used in physics or engineering mechanics, as well as dimensionless integer and real numbers. This permits the explicit integration of engineering rules and formulas into the CAD models. Boolean operators and parameters which can be either "true" or "false" can be used to activate or deactivate design sequences stored in the model. A very useful parameter is of type “string” (alphanumeric) which serves as switch between different configurations. Even measurements stored in the model can be used as dynamic parameters representing geometric characteristics which result from design operations. Finally, geometric elements like curves or surfaces can be used as convenient geometric parameters (see figure 3.2).

Any CAD software requires the user-input of parameters to define the numeric features of individual design steps. Usually, these parameters are hidden inside the model once the geometric definition of the specific design has been accomplished. These hidden parameters are sometimes referred to as implicit or intrinsic parameters. Not only numeric values typed into special user menus but also dimensions defined in sketches and data from external files like tables or associated engineering drawings can be used to create parameters and to link them to the modelled geometry.

To support the comprehensibility of PAD models, all controlling variables and elements must be made clearly visible at all levels of the specification tree. For this purpose, explicit, user-defined parameters are created and displayed in the specification tree (see figure 3.3). To enhance model clarity, they can be grouped in parameter sets. By creating user-defined relations, these parameters are linked to the internal (hidden) design parameters inherent in the model.

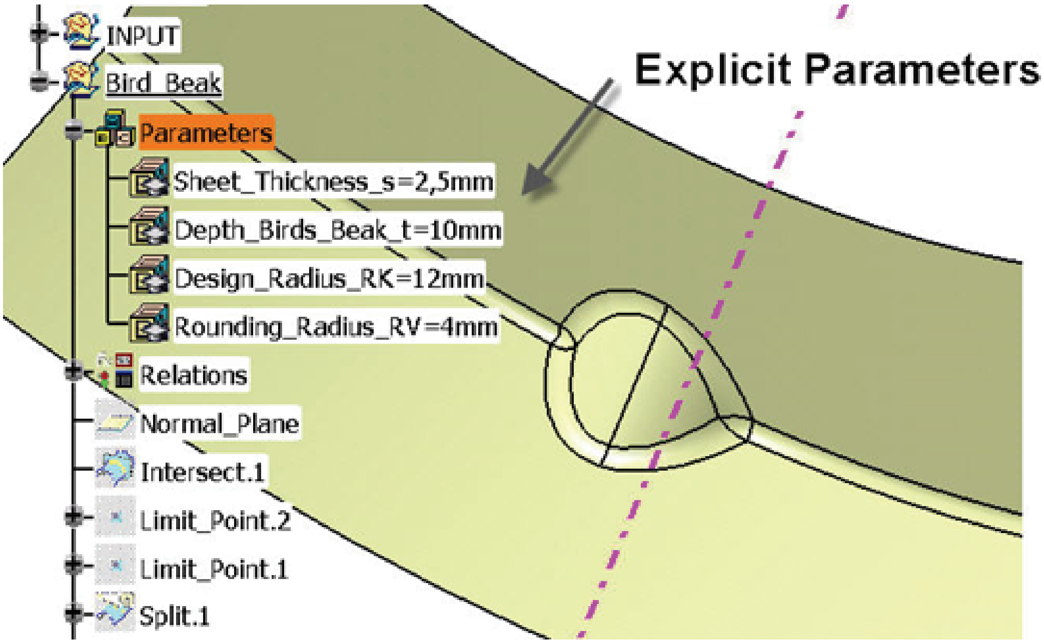


Fig. 3.3 A set of explicit parameters controlling geometric details of a “bird’s beak” (stiffening swage)

3.2.2 Geometric Constraints

Apart from parameters, geometric constraints are used to define geometry in the PAD model. E.g. instead of defining a numerical parameter of type “angle” between two parallel lines and assigning to it the value of zero degrees, the geometrical constraint "parallel to each other" is enforced on the two lines. Other examples are lines which are made coincident to a point or tangential to a curve. With reference to coordinate systems, even constraints like "horizontal" or "vertical" can be assigned to lines.

Among the geometric constraints which modern parametric CAD systems offer are: parallelism, tangency, horizontal and vertical orientation, concentricity, symmetry, equal distance, coincidence, the location of a point in the centre or at the end of a line, the definition of the axes of an ellipse as major or minor axis, etc (see figure3.4).

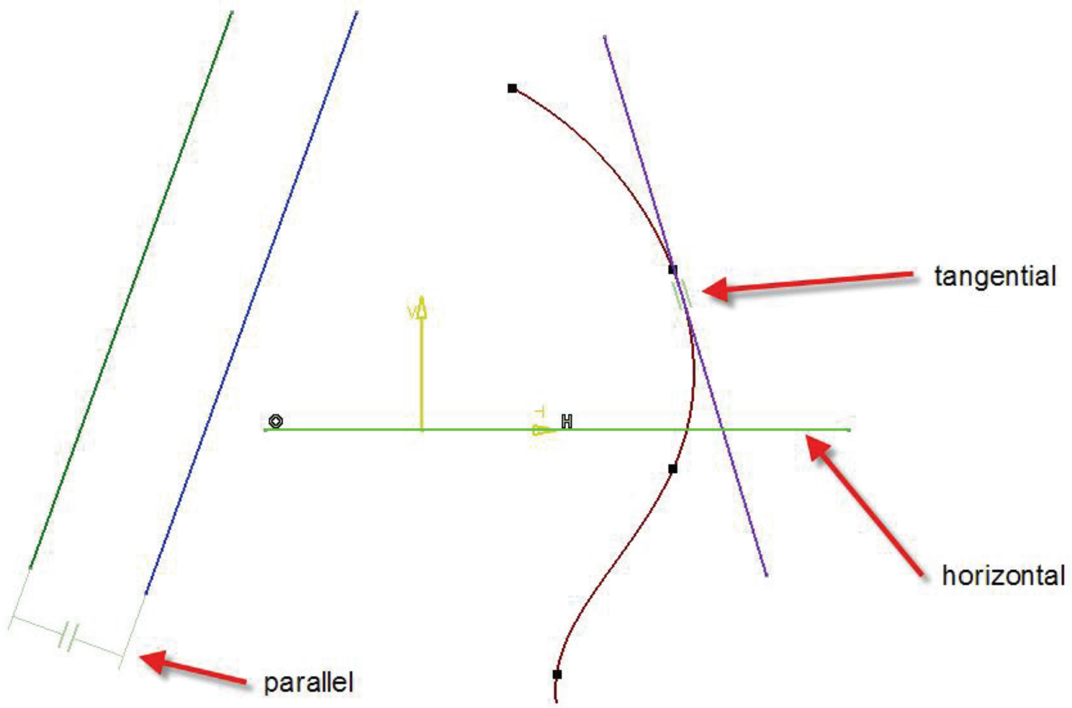


Fig. 3.4 Geometric constraints in a sketch

Comparable to sketches, where the geometric constraints are explicitly represented by small symbols, three-dimensional geometric elements can also be controlled by geometric constraints.

These constraints, whether explicitly displayed or hidden in the model, are an important contribution to the unambiguous definition of design sequences. If one or several of the parameters or geometric constraints are deleted, the definition of the design becomes ambiguous.

3.2.3 Design Tables

Design tables are helpful defining parameter families. A design table permits the assignment of parameters to values stored in an external file which is used for the remote control of parameters in one or several PAD models.

A design table can be created using an existing MS Excel worksheet or a text file containing the names and values of the parameters to be controlled. To associate such an external file with parameters defined in a PAD model, each column (or row) of the file must be mapped to a distinct parameter. If the required parameters have not yet been defined in the PAD model they can be generated from the design table.

However, this method is error prone during the mapping process of parameters to columns or rows of the external file. It is much safer and more convenient to define the parameters in the PAD model before creating the design table. Parametric CAD systems support this method and offer assistance for creating Excel or text files on the basis of explicit parameters which the designer selects, e.g. from the specification tree of the model.

Parameters which are already controlled by a relation or by an associated design table cannot be linked to other design tables or be used to create them, they are not transposable.

Both methods of creating design tables and linking them to the PAD model finally result in external links from the explicit model parameters to the external file. Changes made to the values stored in the external file will affect the PAD model. Both methods permit the later addition of new parameters to the design table.

Design tables are of great use where geometry or part families are to be defined. The use of MS Excel worksheets permits the definition of relations between parameter values stored in one or several design tables.

3.2.4 Relational Checks

A check can be defined to notify the designer of design changes violating predefined requirements. The check uses parameters or geometric characteristics to monitor changes made to the PAD model, and find out whether the changes result in the violation of the specified design restriction. The check does only inform or warn the designer, and it has no effects on the design itself. To assist the designer, the icon representing the check in the specification tree is a traffic light showing a red light when a check is not fulfilled. Where appropriate additional messages or warnings can be displayed to inform the designer of the violated check (see figure 3.5).

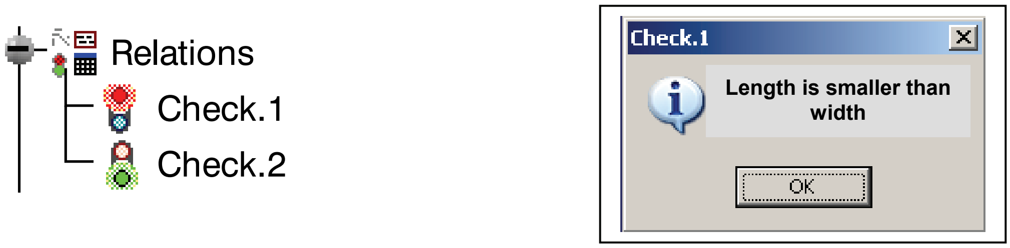


Fig. 3.5 Traffic light application in specification tree and information window with the user defined message "Length is smaller than width"(Braß, 2005)

3.2.5 Formulae

Formulae are used to define parameters or conditions of controlling parameters. They are defined like equations, with the controlled parameter on the left hand side of the equals sign, and the relation defining the parameter on the right hand side of the equation (see figure 3.6).

Parameters as well as the relations associated to them can be imported from external files. It is possible to define several formulae controlling the same parameter; however, only one of these relations may be active at any time.

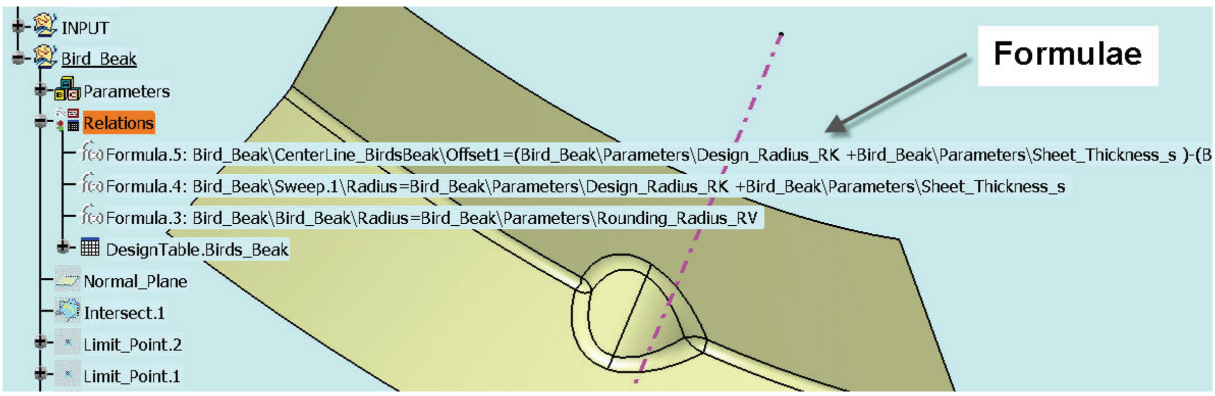


Fig. 3.6 Formulas assigning implicit parameters with values calculated from explicit parameters

3.2.6 Rules

A rule is a simple programmed code controlling the relations between parameters using "If … Then" formalism (see figure 3.7 and 3.8).

Rules can be used to assign discrete values or relations to parameters or to define associations between geometric parameters and the corresponding geometric elements. Rules can become very powerful when they are used to trigger macros coded in a programming language like Basic Script or Visual Basic.

On the contrary the syntax of rules is very limited.

3.2.7 Reaction

A reaction is a rule which is only triggered when a previously defined event occurs. Whereas the rule continuously controls the geometry of the PAD model, the reaction is only activated when the event associated will take place. The list of possible events which may initiate the reaction depends on the design sequence and the features. Different features allow different events to be used to start the reaction.

Reactions offer better control of the points of time when the action programmed in the reaction shall be executed.

3.2.8 Macros

Macros are powerful subroutines coded in Visual Basic for Applications, Visual Basic Script or Basic Script. They can be recorded from operations performed by the designer or programmed from scratch to repeat routine tasks or provide user-friendly interfaces to complex PAD models and design configurations. As macros can control and read explicit and implicit parameters, they can be used to produce protocols of specific design configurations in the form of reports. They can also be applied to pass on parameter values and geometric elements to subsequent steps in a product definition process chain. Typical applications are parameter optimisations with CAE methods and automatic generation of NC program code.

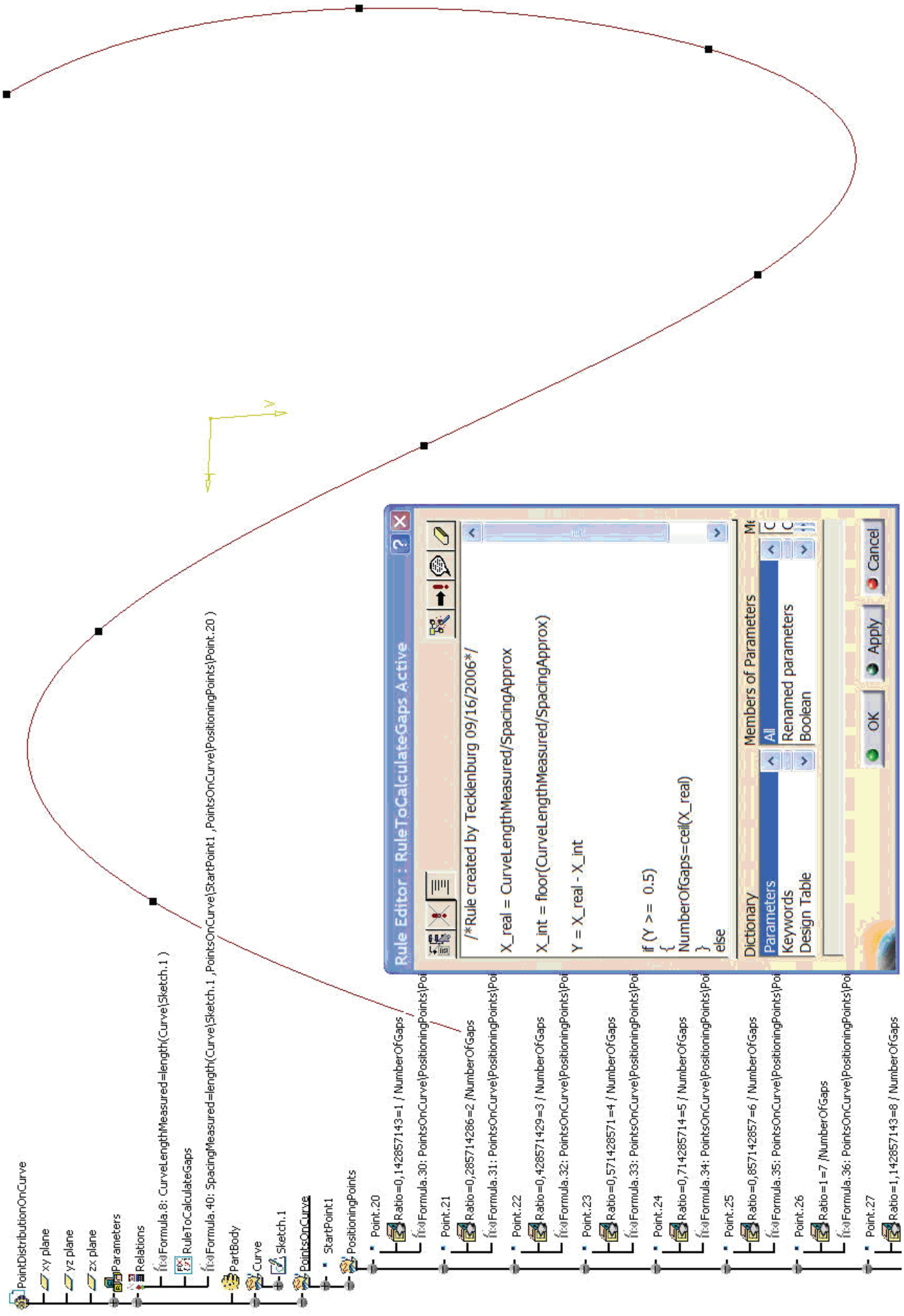


Fig. 3.7 A rule applies a number of points to a spline curve depending on curve length and gap width

However, macros can only reflect the intelligence and the knowledge of the persons involved in their programming. Usually, designers are not well qualified programmers and lack the pertaining experience. They may be experienced in handling parameters and relations or rules but often find it difficult to manage complex tasks of error-handling and designing user-friendly interfaces. On the other hand, professional programmers usually lack the experience of a designer and do not speak the same language. They may not even understand the complexity of the design task. Macros coded by professional programmers will never be accepted and used by experienced designers if they have the feeling that they can no longer use their creativity and intuition in the design process. With increasing subcontracting of complex engineering tasks OEM will lose more and more knowledge and the continuous use of macros will add to this loss and impair innovation.

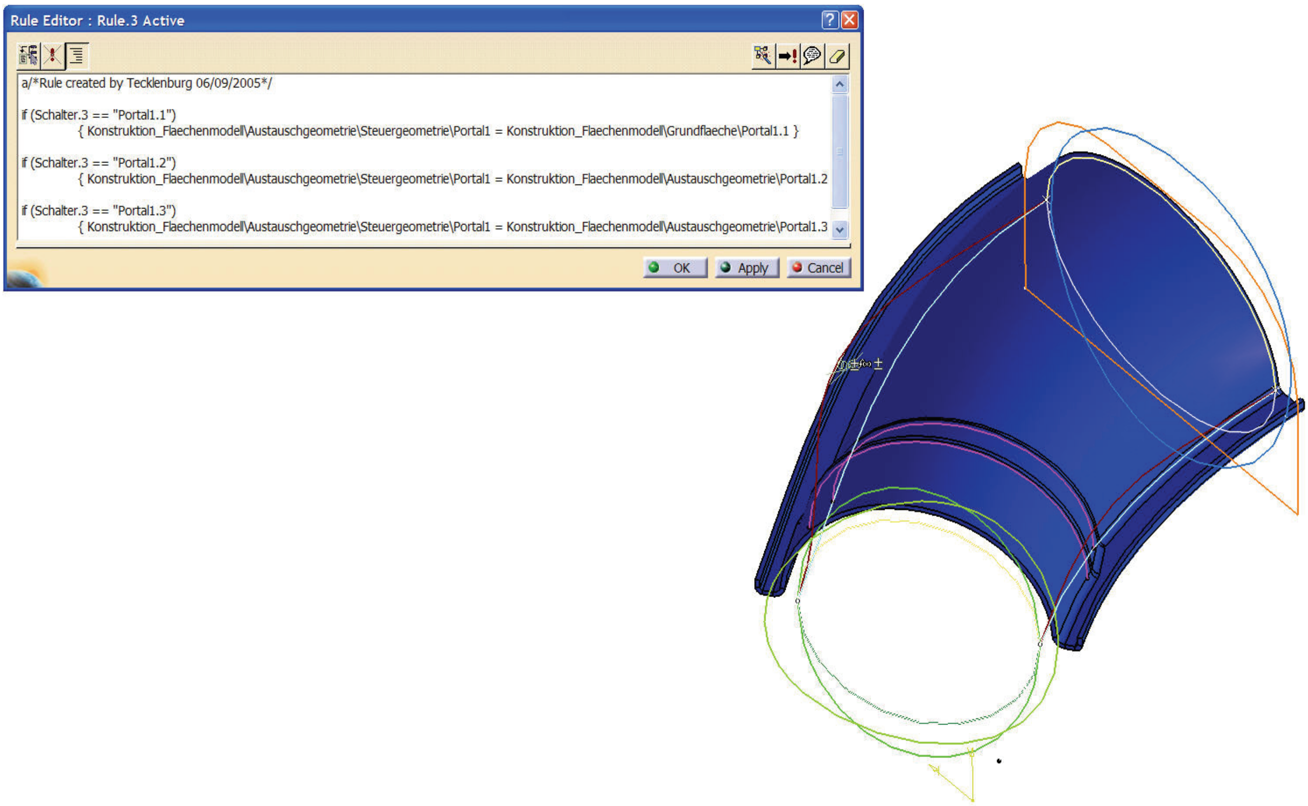


Fig. 3.8 The housing upper shell is controlled by geometric parameters. Parameters of type “string” (switches) and rules allow the designer to choose from 3 portal curves and 2 guide curves (geometric parameters) on either side (see figure 3.2)

3.3 Management of Complexity in PAD models

PAD permits multi-model links (between parts and assemblies) which are used to control the way information is passed on to other PAD models in the process chain, and to initiate resulting changes in these models. Compared with conventional and unparametric models, the file size of PAD models is larger because the complete history of design steps is stored in the model.

To ensure the easier understanding of PAD models, standard procedures must be used for their creation. The more basic geometric elements are required to build a model the less understandable will the model be to persons other than the designer who created it. The designer may not be able to remember his design strategy and might find it difficult to modify the model without causing indefinable update errors or non usable geometry (Brill, 2006).

Consequently, the parametric associative CAD model chain requires a consistent, structured approach with defined interfaces to a relatively large number of small, easily comprehensible PAD part models. This is one of the main differences between PAD and conventional non-parametric CAD modelling.

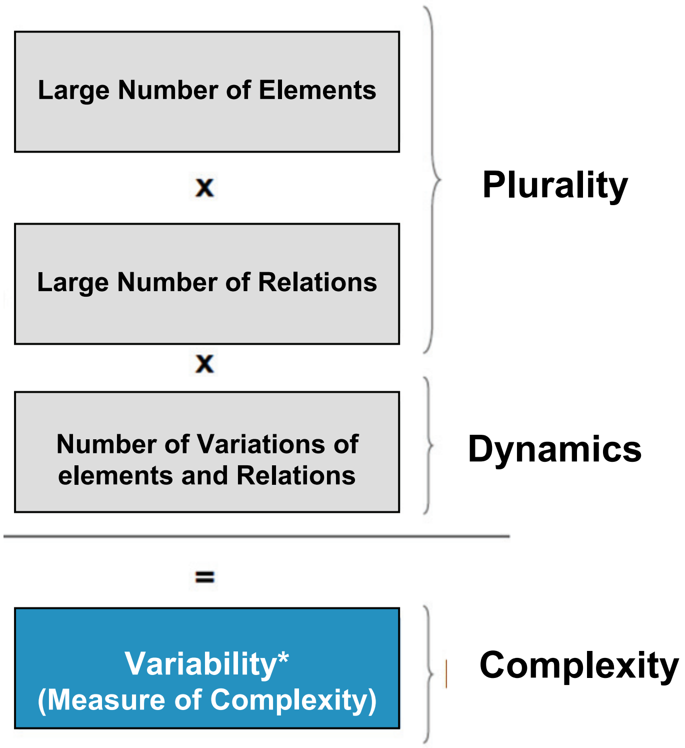


Fig. 3.9 Variability = Number of conditions a system can assume (Wildemann, 2000)

To limit the complexity of a PAD model (see Fig. 3.9), standard specification trees must be used for modelling similar parts (e.g. sheet metal parts, cast and injection moulded parts). Standard modelling procedures must be designed ensuring the consistent application of the Input Design Output (IDO) principle. Uniform naming conventions at all levels of the specification tree on one hand, and a well-structured design method which uses a uniform pattern on the other hand should be employed. This design method starts with defining of primary geometric elements first, followed by secondary geometry and accomplished by details like edge fillets, trimmings and holes or pockets. The number of links can be reduced dramatically, if the individual zones of a design are kept independent of each other for as long as possible. Relations can easier be understood when references are published.

Figure 3.10 shows the IDO principle applied to an assembly in the concept phase. In the input folder, all information required for the design (e.g. Class A surfaces and curves and Carry Over Parts (COP)) are prepared following the IDO structure. From there, the information is released to the actual design process. The required references are inserted into the input folder of the respective part, and processed systematically to generate the desired design geometry. They are then passed on to subsequent processes (with or without active links). In the output folder of the assembly, output information is gathered, structured in sensible sets and published.

The consistent use of the standard structure and standard names ensures that subsequent processes can access the relevant objects in the specification tree automatically, e.g. with the aid of macros.

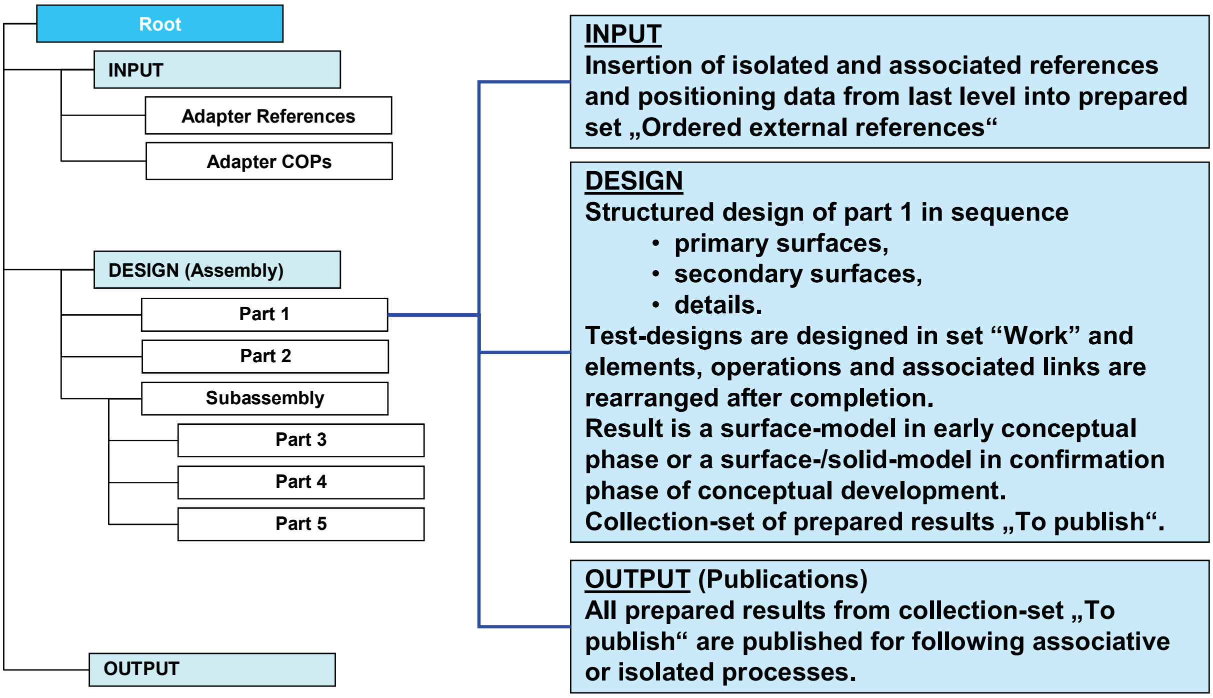


Fig. 3.10 IDO-principle on all levels of an assembly with published references and outputs

Primary geometric elements are often designed sketch-based, using true sections on intersection planes normal to the spine curve of the expected geometry. Each section is resolved into straight and curved elements, which have a curvature exceeding the desired edge fillet radii by far, and which are at least two or three times longer than required, to permit intersecting and filleting between the derived surfaces without update errors. The basic elements are coloured and renamed by the designer to reflect their association to specific parts or processes (see figure 3.11).

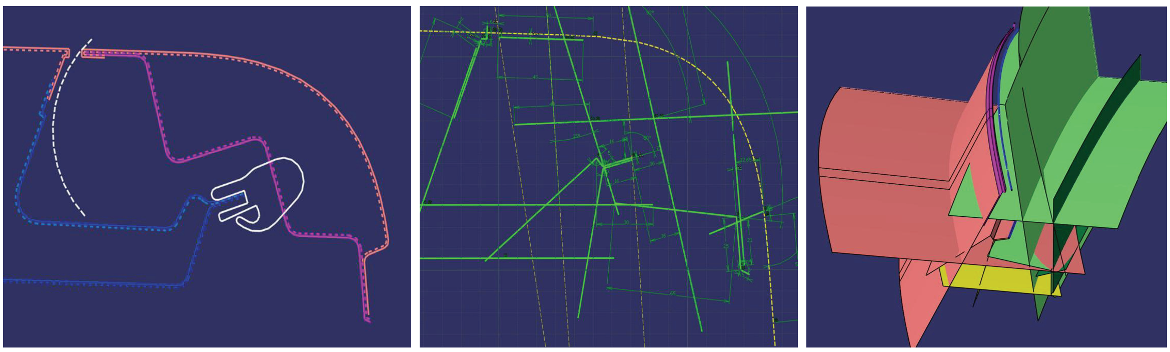


Fig. 3.11 Principle section, true section and primary surfaces derived thereof. Upper area tailgate of a van (Aldag et al, 2006)

Once the boundaries of the primary surfaces have been defined (e.g. through trimming or filleting) the secondary surfaces are designed in a separate process. The examples of secondary surfaces shown in Figure 3.12 are joggles and reinforcements for hinges and the third brake light. When the curves and surfaces are created, the unambiguous definition of their directions is of great significance because subsequent operations like trimming, filleting and offsets depend on the direction of the original elements.

The design work is performed independently for the various functional zones into which the part or assembly has been subdivided (e.g. tailgate upper, side, lower, hinge attachment, lock attachment etc.). The geometric elements generated for the individual functional zones are kept separate from those of other zones as far as possible to enhance the comprehensibility of the PAD model and keep the links clearly arranged and the number of links to a minimum.

It is the purpose of this thesis to apply the proven method of collaborative manual design on large design tables to the methodical creation of PAD models, permitting complete assemblies to be designed simultaneously in a zone-based approach.

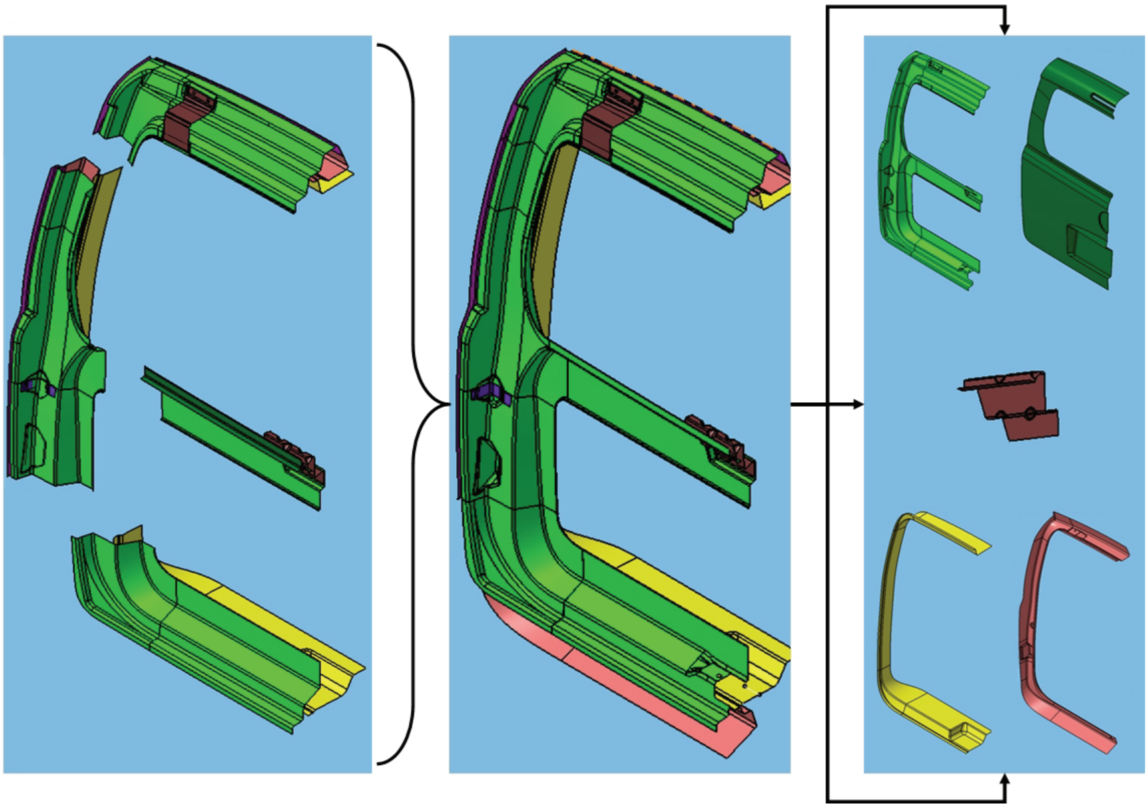


Fig. 3.12 Zone-mode design of a tailgate and its corresponding surfaces (Aldag et al, 2006)

Only after all design details such as joggles, edge fillets, holes have been created, the zone models are combined to make up the complete assembly and derive all individual parts. When using this design method, it is advisable not to design the specific assembly (e.g. a tailgate) in an isolated manner. Instead, the neighbouring surfaces of adjacent parts (in this case: roof, roof rail, side panel, bumper) should be modelled within the same process.

Another method for generating the design of individual parts from e.g. the zone based concept design model is the manufacturing-based approach, which utilises the individual manufacturing process steps to structure the design. For a deep-drawn part this would be: drawing step 1, drawing step 2, trimming and punching operations. For a cast part this would be: unmachined cast, milling and boring operations. This method permits the easy reuse of the PAD models in production planning processes but requires early communication between product engineers and production process engineers.

3.4 Dependencies and Associations

The key requirements to be met by a parametric associative CAD model are the unambiguous and complete definition of the geometry (integrity) and the sensible interrelations (model coherence).

The central idea of associative design is that each design element is generated in a geometrical, technical or functional context or is related to such a context. The term "associative design" is derived from the verb "*to associate"* (Leo-Dictionary, 2007) conveying the concept of recording and modelling the dependencies of a geometric element of its environment (Forsen, 2003).

As several different types of links can be established in a PAD model, the various link types are explained in detail in the next paragraphs:

3.4.1 Geometric Constraints

Most CAD designs are based on sketches describing the two-dimensional geometry of the cross-section of a part or an assembly.

Conventional drawings depict the coordinates and dimensions of geometric elements used to make up the design. Geometric constraints like parallel, perpendicular, coincident etc. can be used to describe the geometric relationship between the individual elements (see 3.2.2).

In addition to 2D constraints used in CAD sketches, PAD systems support the definition of three-dimensional geometric constraints (assembly constraints, instance-to-instance links) for creating assembly designs from existing parts (bottom-up method) and the definition of kinematic constraints required for simulating the kinematical behaviour of an assembly .

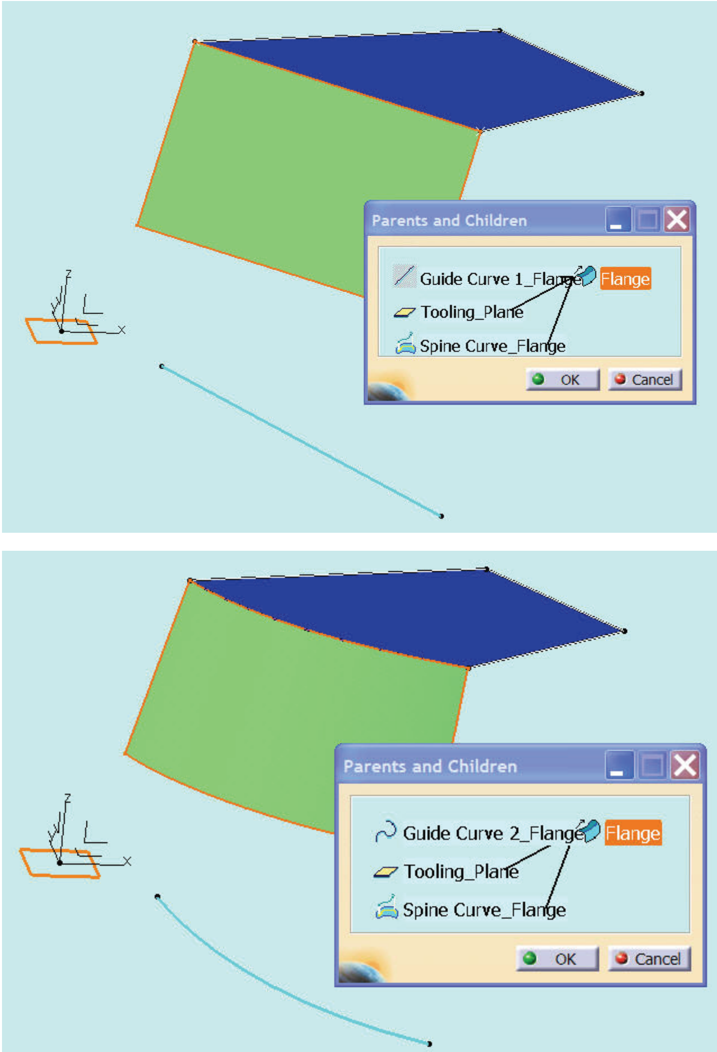


Fig. 3.13 Parent/child relations in the example of a swept surface (child)

3.4.2 Parents/Child Relations

To aid designers in controlling the associations they establish, PAD systems can display so-called parents/child relations of individual design steps. Upon the replacement of parameters or geometric elements (parents), the child elements are automatically adapted to reflect the changes provided they permit to unambiguous recreation of the child geometry (see figure 3.13).

3.4.3 Multi-Model Links (MML)

PAD systems permit the unambiguous definition of parametric associative assemblies based on links between individual parts and assemblies defined within or between product contexts. They are directed links, pointing from the receiver model (dependent part) to its sender model (reference). In most cases, these links are established between geometric objects. In CATIA, such links are referred to as multi-model links (see figure 3.14, Braß, 2005).

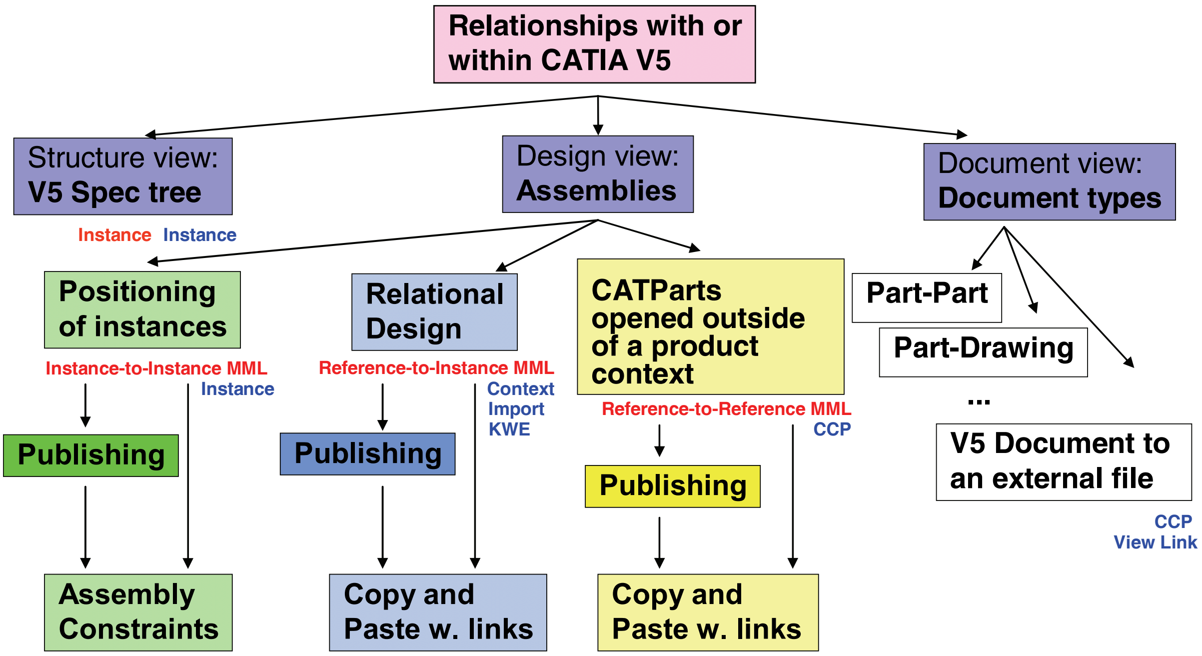


Fig. 3.14 Overview of multi-model links (MML) in CATIA V5 (Braß, 2005)

Below, some examples are given of multi-model links and their handling:

3.4.3.1 Instance Link

An assembly can comprise several identical parts. When modelling such an assembly with a CAD system the file containing the geometric information of the repeated part is only loaded once (as the original part) into the assembly model. Subsequent additions of the same part to the assembly lead to the creation of further instances of the same part, with all instances of this part pointing to the same original file. To distinguish the instances, they are usually identified by instance numbers (see figure 3.15).

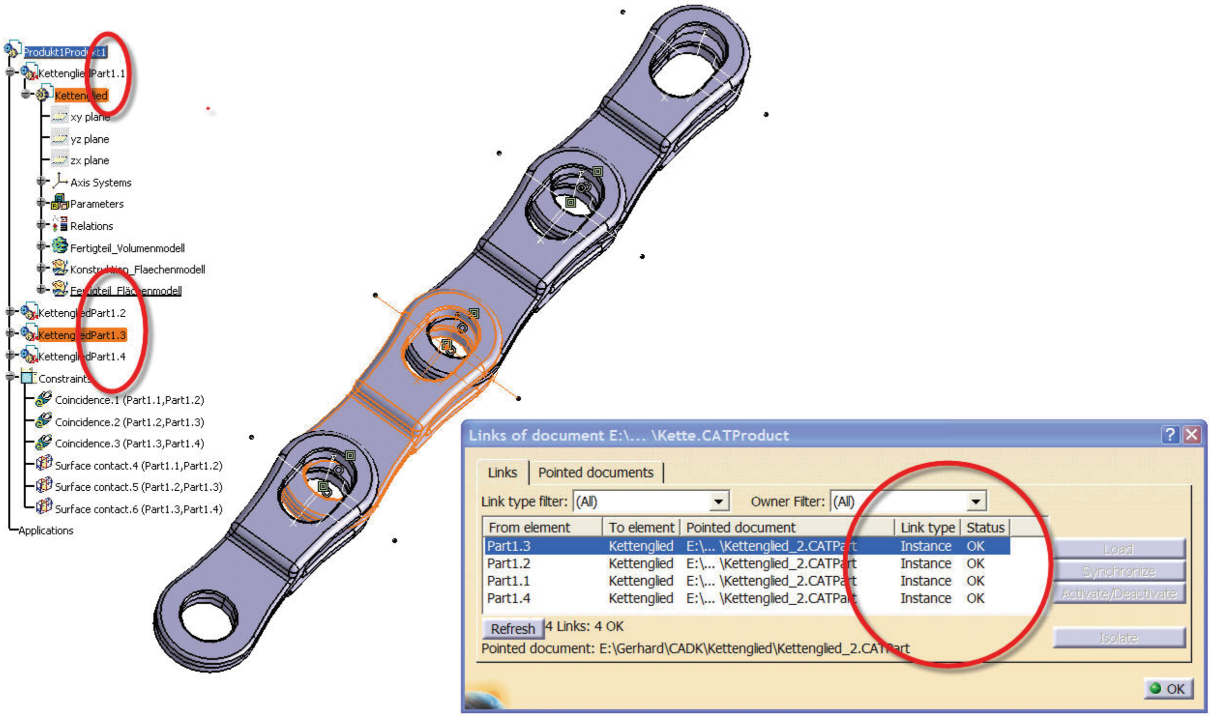


Fig. 3.15 There are four instances of the same part in the assembly: four instance links point to one document.

3.4.3.2 Instance-to-Instance Link

When individual parts and subassemblies are joined together to define an assembly with the bottom-up method using assembly constraints (3D geometric constraints), these constraints cause links to be established between an instance of a part and an instance of a different part or a different instance of the same part (see figure 3.16). The unambiguous and complete definition of all required assembly constraints to define the desired degree of freedom for the parts is only possible by manipulating the individual instances of the parts, taking into account the constraints which have already been defined.

3.4.3.3 Reference-to-Instance Link

Assemblies containing sub-assemblies and individual parts which have been created according the top-down principle (according to the method of design-in-context / relational design) contain special folders (skeleton, adapter, mating part) to control the references at each hierarchical level of the product structure. The links between these folders must all point in the same direction. Generally their direction is vertical, with horizontal links only permitted inside subassemblies. The behaviour of the links depends on the context in which they have been defined.

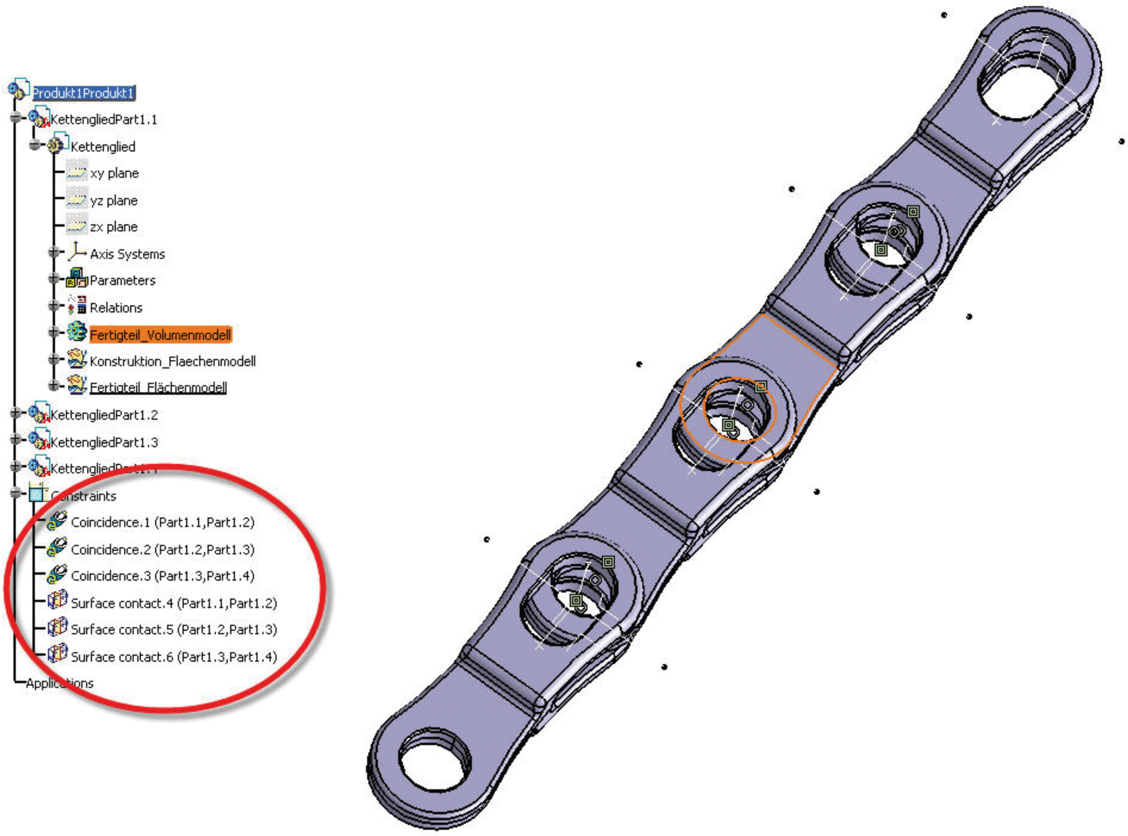


Fig. 3.16 In the assembly shown above, a geometric constraint points from one instance of a document to another instance of the same document.

Links which are established between parts which are defined in the same product structure and in one window are referred to as Reference-to-Instance Links (see figure 3.17). So-called Import Links point from the dependent part to the product node. Additionally, a so-called Context Link is created pointing from the product node back to the dependent part. The Reference-to-Instance Link is thus based on the relative path between the documents of the assembly. The dependent part knows its dependency of the assembly and at the same time the assembly knows its dependent part.

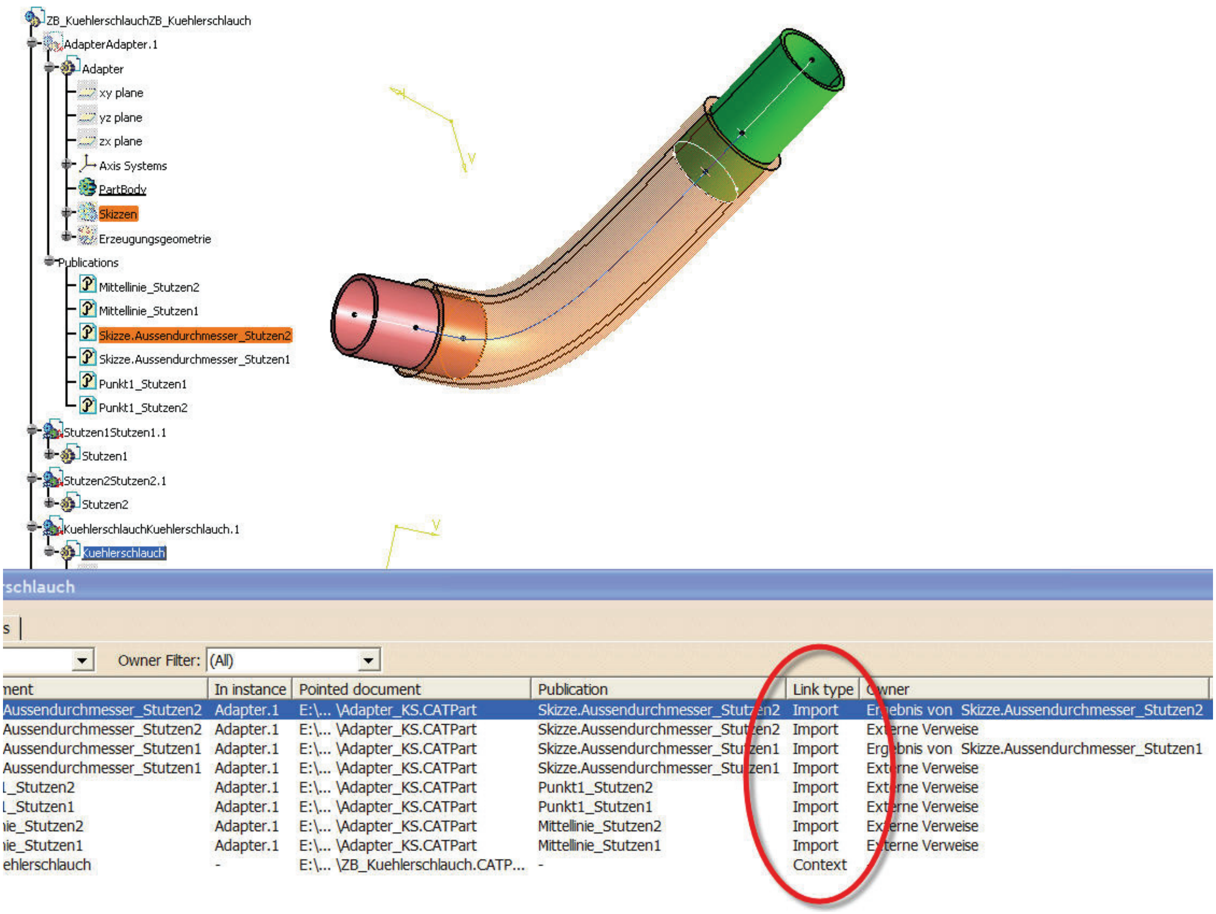


Fig. 3.17 Reference-to-Instance links in an assembly

If the references on an assembly level are modified or exchanged without editing or deleting the links, the dependent design is automatically modified to reflect the changes. These links also take into account the positions matrix of the part. If the position of a part is defined relative to a reference part and the reference part is moved, the dependent part is repositioned automatically. On the other hand, the dependent part cannot be moved; it requires the reference part to move it.

In complex assemblies, it is advisable to control the effects of design changes on dependent geometry using a combination of view links and manual links.

3.4.3.4 Reference-to-Reference Link

References established between parts which are not opened in the product context but in separate windows (see figure 3.18), are created as Reference-to-Reference links (in CATIA they are called Cut Copy Paste (CCP) links). This type of link is defined under support of the element names and the Unique Universal Identifiers (UUIDs) (see glossary) of the respective documents and is only active if the original documents (with the UUID numbers stored in the link) between which the link was established are loaded into the CAD system simultaneously. As no document paths are stored in the link, the pertaining part documents cannot be loaded automatically.

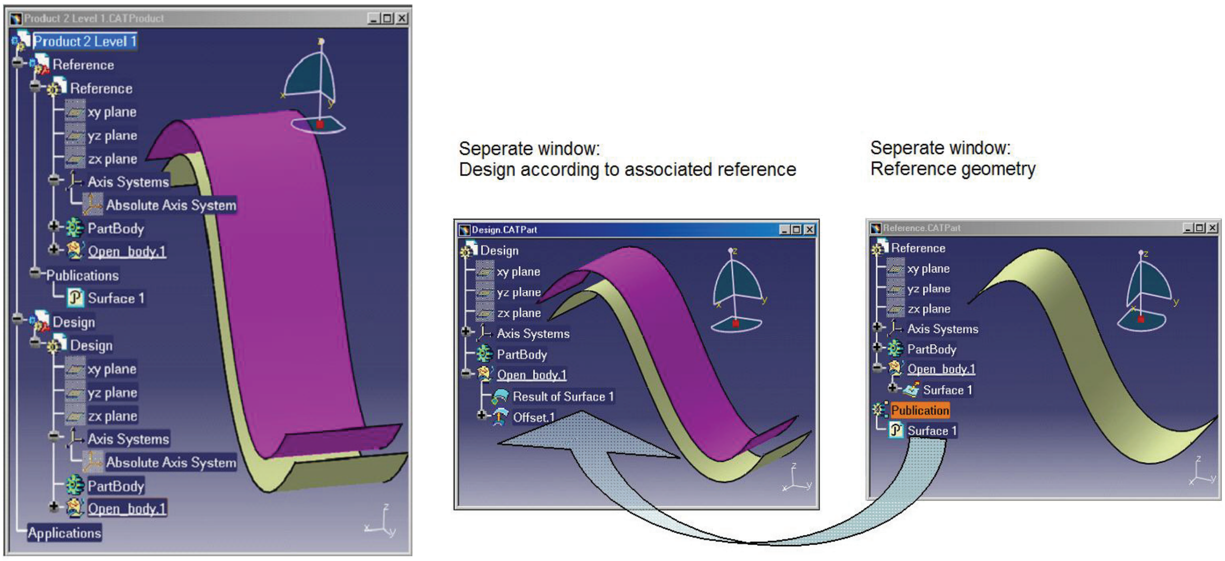


Fig. 3.18 Update behaviour with Reference-to-Reference Links: the change of the position of the reference part does not change the position of the dependant part (Braß, 2005).

The dependent part knows that it contains a link to the reference part, but the reference part does not know any of the parts it is linked to. This type of link does not contain any information relating to the position matrix of the dependent part. If the reference part is moved, the dependent part does not move automatically. Consequently, it is possible to move the dependent part without moving the reference part.

3.4.4 Management of Associative Documents

3.4.4.1 Handling the References to Control PAD Models

To handle multi-model references in PAD models, structuring elements like specific geometrical sets (CATIA), parts or assemblies are used to establish a management system for the reference geometry which controls linked parts and assemblies.

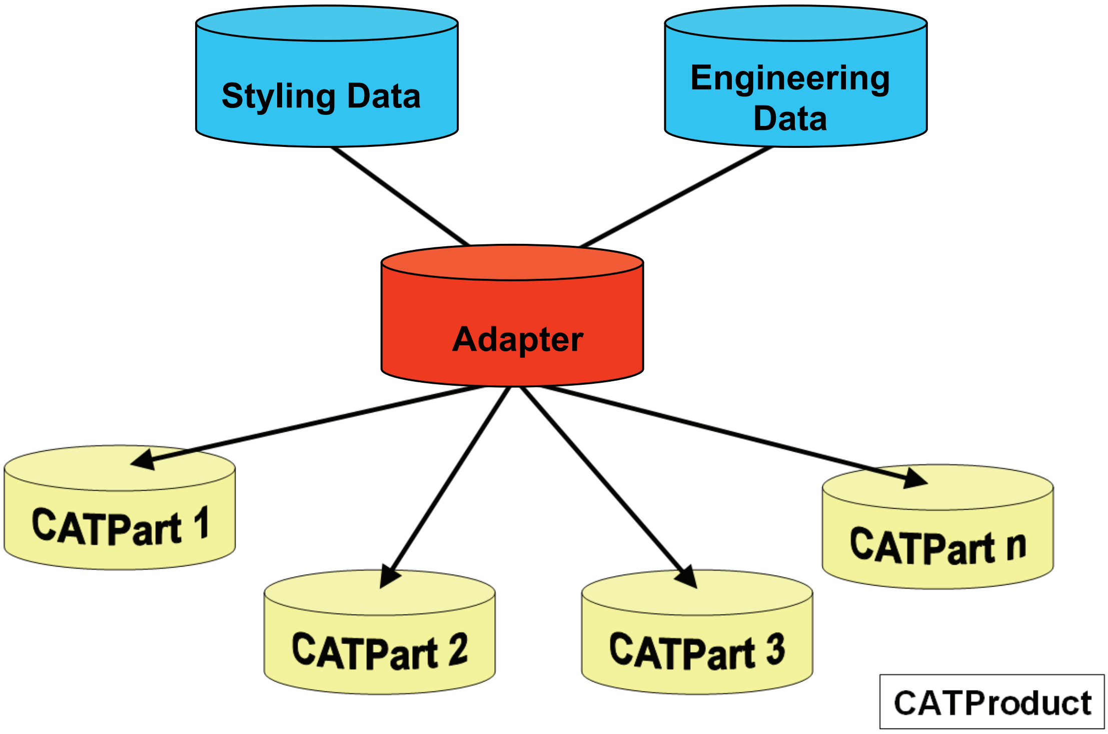


Fig. 3.19 PAD assembly with four parts and one adapter which hosts control parameters and reference geometries from previous process steps (Braß, 2005)

In the context of a complex parametric associative product structure, e.g. a Body-In-White (BIW), these management systems serve as interfaces between the individual part and assembly designs of the product as well as between the individual phases of the development process. They are therefore the key elements for the systematic structuring of a parametric associative product development process.

The literature describing such management systems uses various names for the elements used as interfaces. They are called skeleton, adapter, interface, reference or mating part models (see figure 3.19). The definitions are often used as synonyms without considering differences which may exist between the various subtypes of management systems. Forsen (2003) defines "mating system" as superior to the systems "adapter" and "skeleton". According to Forsen, the skeleton system does not establish links to designs outside the context of the currently defined design, whereas the adapter system is capable of establishing links to any design irrespective of the context. Forsen describes the skeleton system as defining fundamental internal geometric structures and links in an isolated PAD model, contrary to the adapter system serving as the interface to corresponding assemblies. Forsen stipulates that the skeleton system can be considered as a special subtype of the more general adapter system.

The name skeleton system is derived from the noun skeleton[1], which stands for the main structure supporting the design. The skeleton model of a PAD, controls basic geometric elements and structures with respect to their dimension, orientation and location in the space. Depending on the hierarchical level, the skeleton model controls the location of individual geometric elements or complete parts. In automotive body design, it is common practice to create grid-sections (relative to vehicle coordinate system) at previously agreed locations. These grid-sections intersect the design results and permit the validation of package spaces besides DMU. The location of the agreed intersecting planes for example can be defined in skeleton models and then be made available to the entire process.

The most frequently used term for the link management systems is the term "adapter". It is derived from the verb "to adapt“[2]. This term depicts the intention of the management system: to collect references in a comprehensible manner at the interface between superior and subordinate parametric associative designs and make them available to the design process, and to support their adaptation to modifications of the design. Additionally, the adapter system can be understood as a system for subdividing complex linked assembly designs into comprehensible and manageable ones. Thus adapter models are suitable for keeping the parametric associative design of subassemblies and package space independent of each other while taken the mutual geometric interrelations into account (see figure 3.20). They help to keep the number of links to a minimum and ensure comprehensibility of the links established between all components involved in the design. The adaptability of a PAD depends on the quality of the references defined in the adapter and the stability and robustness of the parametric relations.

[1] skeleton = the main structure that supports a building, etc. OXFORD (2000)

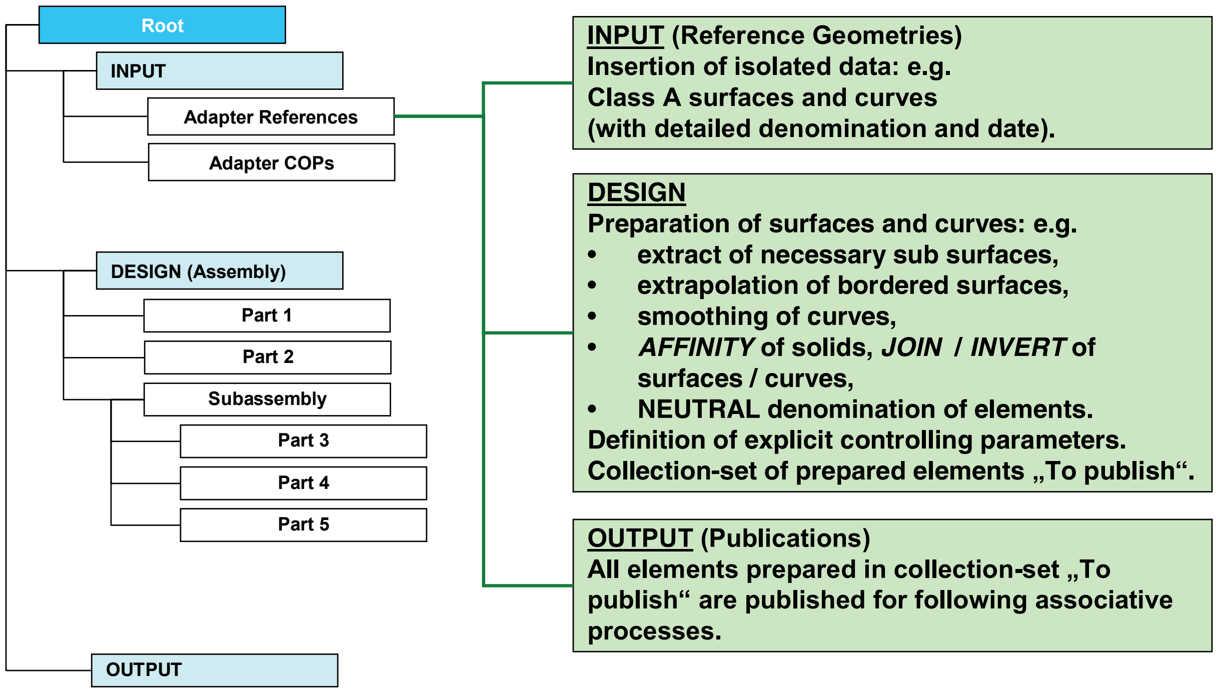


Fig. 3.20 Systematic Preparation of References in the Adapter Model on the Example of design Surfaces and Curves

Haslauer (2006) compares the adapter system to an adaptable plug system used for electric connections. He believes that existing and newly established reference elements should be used to permit the adaptation of PAD model via the adapter interface.

Inside an assembly hierarchy containing several subassemblies, the management systems for reference geometry and parameters are always hosted at the top assembly level, and they point downwards to the direction of the dependent subassembly levels and parts.

Within a complex multi-assembly PAD model, several separate adapters can exist at the various hierarchical levels inside the model. Additionally, different adapters can be defined at the same hierarchical level to separate different categories of input information. Inside the adapter model, the input geometry is processed to make it "adaptable" for the geometric operations depending on the reference geometry, e.g.

[2] adapt = sth (for sth) to change sth in order to make it suitable for a new use or situation OXFORD (2000)

by extracting surface sections, smoothing guide curves, re-orienting surfaces and curves, renaming elements according to naming conventions and publishing them. Quite often, the initial creation of a PAD model is made in the early stages of a product development process before the required input data is available. In this case, the interface models are defined with dummy geometric elements of the same element type as the expected real input geometry. This dummy geometry is then used to build the PAD model, and later in the design process it is replaced by the correct reference geometry. A high-quality PAD model can then be updated and evaluated to reflect the current state of the actual design project.

3.4.4.2 Vertical and Horizontal Dependencies

To maintain the clarity of the association network created between all geometric elements and to ensure stable update processes, it is advisable to define links between parts and subassemblies on the next higher hierarchical level of the model i.e. vertical references.

The definition of direct horizontal references between different adapters or parts which belong to the same hierarchical level is not advisable because such links are hardly comprehensible and bear the risk of closed-loop links (e.g. two parts with cross-references to each other which cannot be evaluated). Such cyclic links cause undesired interruptions when the model geometry is updated.

On the contrary in the detailing phase of a design is sometimes difficulty to define links to hierarchical levels far away. Therefore in some cases it is more suitable to define single horizontal links.

3.4.4.3 Links in Complex Assembly Structures

The association network of complex assemblies makes it difficult to identify the root causes of update errors after design changes or the replacement of geometric elements. In such complex models, view links should be used to visualize the changes before the manual links are modified to adopt the changes.

In the adapter at the next higher hierarchical level, an automatic as well as a manual (isolated) link are defined for the same geometrical element (see figure 3.21).

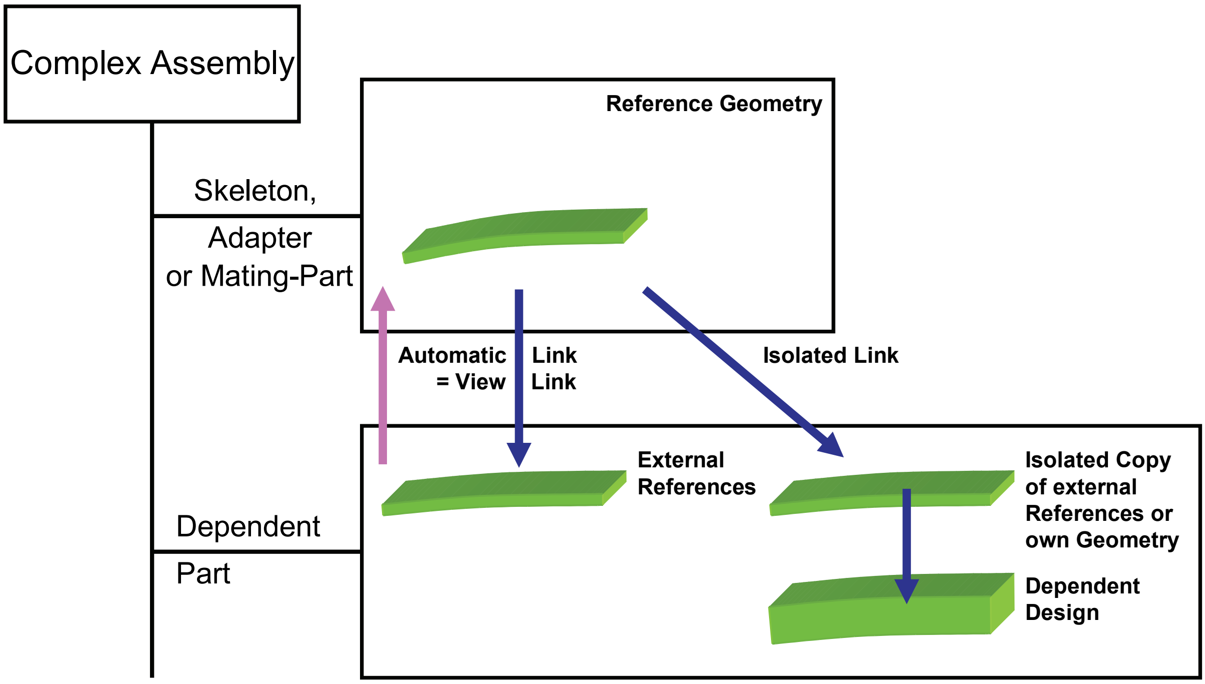


Fig. 3.21 Definition of view links and manual links

The automatic link is copied into the design geometry but not used for design operations. Instead, the isolated manual link is used to build the design geometry. The automatic link always visualizes the actual geometry of the reference element without influencing the geometry of the dependent element, this is why it is called "view link". The designer evaluates any changes made to the reference element and their impact on the geometry defined in his model. Only when he is sure that the changes will not cause update errors or other problems, he replaces the outdated, isolated manual link with an isolated copy of the modified reference element to update his geometry and adapt it to the changes. Thus there is no automatic update of his model, and it is the responsibility of the designer to check the view links on a regular basis. On the other hand, he never risks that his model is destroyed by update errors which cannot be resolved. He always has the time to guard his model before exploring the consequences of any design changes.

3.5 Useful Steps for the Implementation of Parametric Associative Design

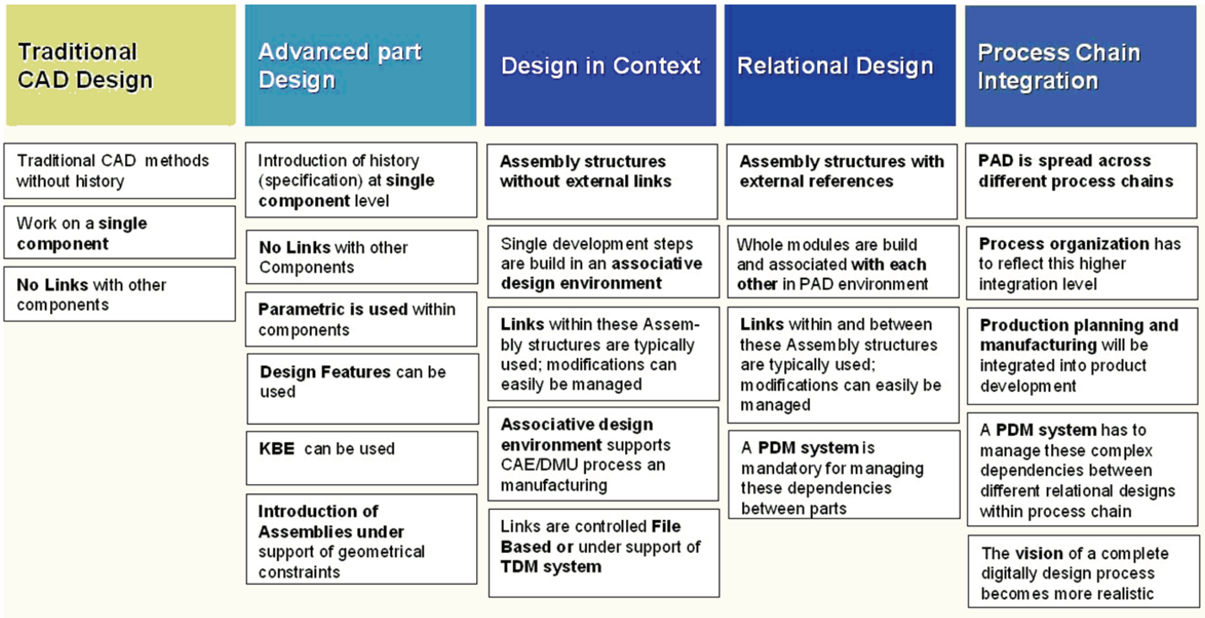


Fig. 3.22 Author’s interpretation for the implementation steps needed for PAD (sources: BMW, DC, IBM)

The complexity of modern automotive development and their implementation in an even more complex parametric associative process requires a careful introduction of all persons (employed by OEM's and subcontractors) involved in the process to this new concept. The effort for changing the design process from the conventional, non-parametric approach to the new principle of consistent PAD can be compared to the effort which was required when moving from manual drawings to conventional computer-aided design (see figure 3.22 and the following explanations).

3.5.1 Advanced Part Design

To implement PAD successfully, the designers should learn to define and use parameters and associative links within a single part. Their organisations should develop and supply standard methods for the structured development of surface, volume and hybrid design models.

Comparable to the philosophy of reusing standard components in manufacturing (so-called Carry Over Parts (COP)), the organisation should define standard specification trees in start models used for the specific design tasks. Equally, standard methods should be defined for creating design features. Templates of drawing frames and standard texts or symbols should be defined to speed up routine processes and support uniformity of the design. Such templates could be stored in catalogues making them easily available to other designers.

Additionally, the organisations should define detailed procedures for establishing links within parametric parts and for integrating engineering knowledge in the models or in the CAD system. Often, tools for integrating knowledge are referred to as Knowledgeware. They permit the definition of rules and other relations between objects defined in the PAD models. Knowledgeware helps to store the knowledge of an organisation and its employees in such a way that it can be reused at a later point of time.

As this unsophisticated implementation step does not allow any links between parts, product design is performed following the bottom-up principle. All parts are defined independently of each other and then arranged together with assembly constraints to define the product structure. The conventional file-based method of storing CAD documents does not cause any problems with PAD files which do not have any external links.

Among the German automotive manufacturers, Daimler was the first to develop standard procedures for this implementation step of PAD in individual models, and have maintained in this step. Their concepts have been adopted by many other organisations when developing methods for parametric associative design (Daimler, 2006), (Brill, 2006).

3.5.2 Design in Context

The implementation step "Design in Context" allows the controlled establishment of links between parts belonging to the same subassembly. However, no external links are allowed to point from the subassembly to other PAD models. Design in context is the first step towards parametric associative design of product structures.

It adopts the principles of advanced part design and offers a structured approach to managing the simultaneous design of individual parts within the context of a product structure.

As opposed to advanced part design, product structures created with the design in context method contain several part models (adapters) which host controlling parameters and reference geometries used in the design of several dependent parts. The adapters are always defined on the product level of the specification tree, ensuring the vertical direction of the links to the parts embedded in subordinate components (assembly, sub-product, and subassembly).

Whereas changes to isolated PAD parts have no effect to the environment of these parts, the modification of controlling parameters in models established with the design in context method invokes changes in the individual part as well as its environment.

Standardized well-structured product models can contain output information for subsequent process steps like DMU kinematics, structural analysis, tolerance simulation and generation of manufacturing plans and programs. These output data are automatically updated when the underlying geometry is changed.

The storage of assemblies containing links between individual models requires special care to ensure that the links can always be evaluated and resolved. If files are stored on data carriers in a file-based way, the save management functionality of the CAD system should be used to assist the process of saving the individual files. Alternatively, team data or product data management systems can be used to manage the storage of complex PAD models.

The design in context method is the suitable method for the design of assembly groups developed by one engineer or a small group of engineers throughout one process step (e.g. concept phase). The method can be performed with Single Model

Links (SML), where the assembly is defined in a single part model using geometrical sets (folders) to describe sub assemblies and detail parts. The second way to apply this method is the use of Multi Model Links (e.g. Reference-to-Instance-Links) where the assemblies and subassemblies are defined in product models and the detail parts in part models.

3.5.3 Relational Design

Encompassing the two implementation steps described above, the method of relational design realizes the establishment and management of links and relations between assemblies and their development processes.

The file-based storage of linked assemblies and development steps is extremely difficult to manage because of the high number of individual model files involved and the distribution of design responsibilities in various departments. To coordinate distributed design tasks and responsibilities and to manage a growing number of design variants and versions, TDM or even PDM systems are required.

3.5.4 Process Chain Integration

Parallel to the efforts of implementing parametric associative methods in the car body design, the implementation of theses methods in other design processes and e.g. in production planning and tool design are being investigated.
The long-term objective is the stable establishment of all required inter-model relations and links throughout the entire process chain of product definition and manufacturing.

3.6 Summary

This chapter describes the change of CAD from conventional non-parametric design to parametric associative design. Due to the complexity of relations and associations established in PAD models, this new design approach requires a new disciplined and structured methodology among designers.

The applications of the different links described strongly depend on the Product Data Management (PDM) system of the OEM/supplier which must be able to administrate the links. In the German motor industry this situation led to different results. In the early phases of product development where the single engineer is responsible for a large assembly or module two different methods are in use:

1) BMW introduced a single model method. In this method from large assemblies to the complete Body in White (BIW) all assemblies, sub assemblies and parts are designed in a one part model. The parts, sub assemblies and assemblies are stored in geometrical sets (folders). This method is only possible in wire frame and surface design and uses in part links for the associations between geometries and parameters. To keep the file size small only primary surfaces can be designed.

2) When it is necessary to design secondary surfaces and details as well as solids the file based method based on Multi Model Links (MML) is used in the early phases. According to part or assembly character parts are stored in part models and sub assemblies and assemblies are stored in product models. The whole assembly of sub assemblies and detail parts is defined in a product model. Reference-to-Instance links (import and context links) are used to administrate the relations between geometries and parameters.

As soon as different engineers, departments or companies are involved in the development of an assembly and in the detailing phase of product development where assemblies are often split between different engineers the MML method under support of Reference-to-Reference links (CCP-links) is used. In this method again the whole assembly of sub assemblies and detail parts is defined in a product model.
But in contradiction to the first MML method described above the design of the single sub assemblies and parts of the assembly takes place on different computers worked out by different engineers.

The author has experimented with all three methods as well as the simple method using Instance-to-Instance links (assembly constraints). As PAD offers its most advantages in the early phases of development where lots of variants are approved and several changes appear the author decided to work with MML method using Reference-to-Instance links.

The implementation of well-structured methods for design in the assembly context permit the enrichment of PAD models with information for subsequent product development steps and the re-use of models. Before this new design approach can be applied successfully to car body design, the following questions must be answered:

What are the differences between the car body design process and other product definition processes in the automotive engineering sector? (See chapter 5)

Which modelling methods are suitable and applicable to car body design? (See last chapters where applications of PAD are described)

How are the modelling methods implemented in parametric associative design processes?
(See last chapters where applications of PAD are described)

The following chapters attempt to give answers in more detail to the above questions.

Chapter 4 Principles of Methodical Development and Design

A product life cycle (see figure 4.1) consists of seven major phases: product planning, product development, production planning, production, sales, product use and disposal/recycling. It is a complex, iterative and multi-disciplinary network of processes, of which the first three phases (from product planning to production planning) are generally summarized as the Product Evolution (formation) Process (PEP). In this process the product design plays a key role (Liese, 2003).

The competitive production of technical products strongly depends on the capability and the performance of the development and design process. This process comprises numerous individual and interrelated tasks which must be accomplished (or rather: solved), and it is influenced by company-specific constraints as well as market-specific and organisational development trends (according to directive 2221 of the association of German engineers (VDI directive)).

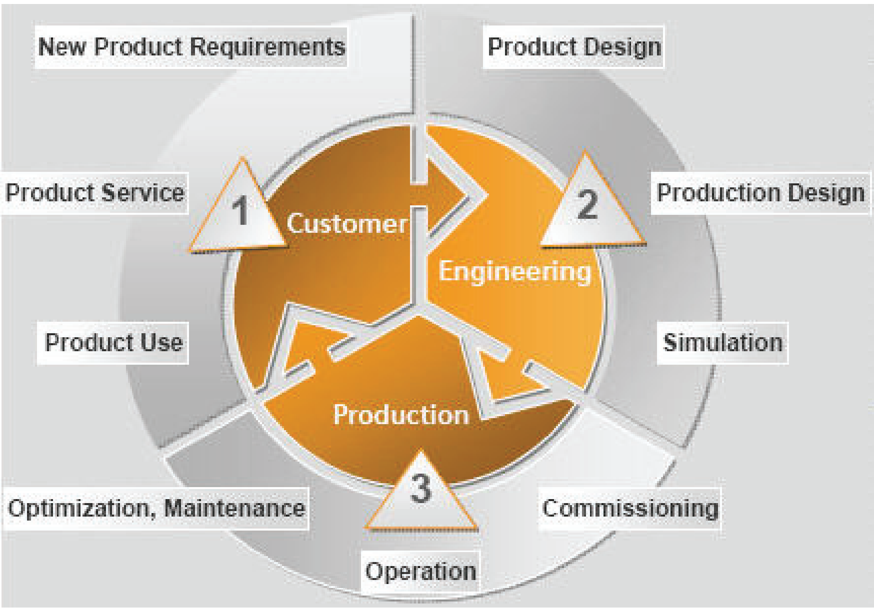


Fig. 4.1 Seamless product life cycle (Schlögl, 2007)

As outlined in VDI directive 2235, value-analysis projects have shown that the costs of a product are mainly determined in the design phase, whereas later phases in the product realisation process are only of minor influence (see figure 4.2). The product costs considered in the analyses do not only comprise the manufacturing costs but rather the total costs caused by the product (i.e. costs of its development, manufacturing, use and disposal). As can be seen in figure 4.2, the development and design processes determine approximately 70% of the total costs of a product. However, the costs of these initial processes amount to only 8% of the total costs.

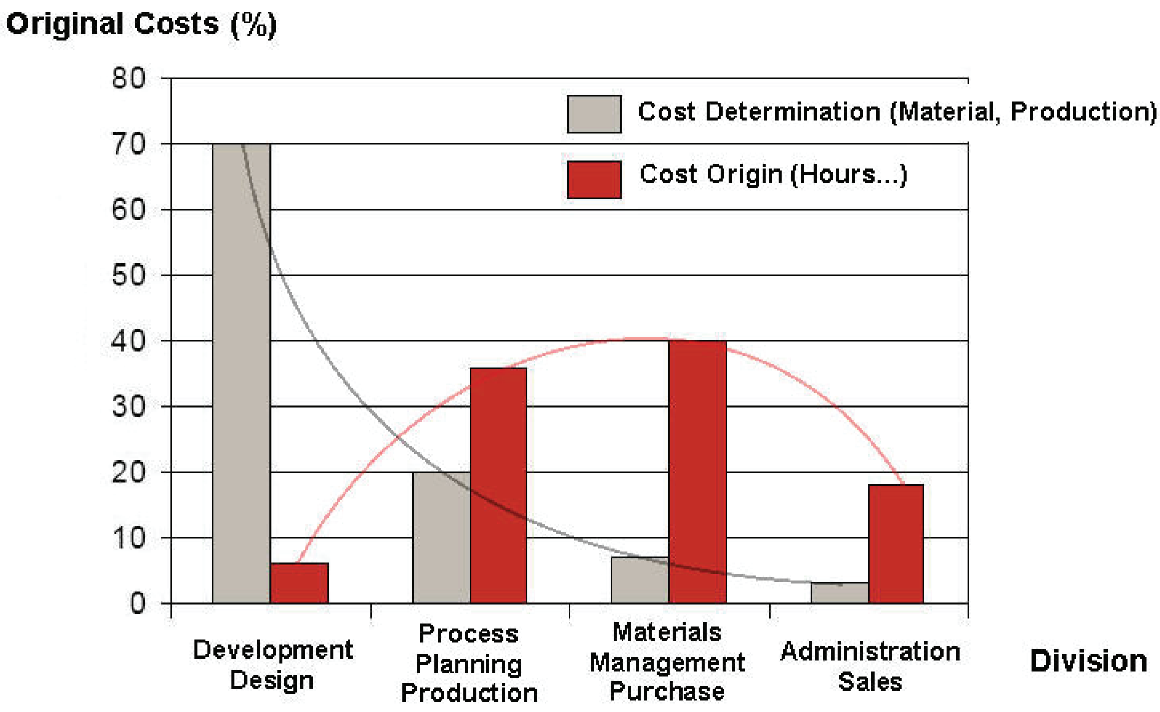


Fig. 4.2 Cost determination and cost origins within company divisions (VDI-Directive 2235, 1987)

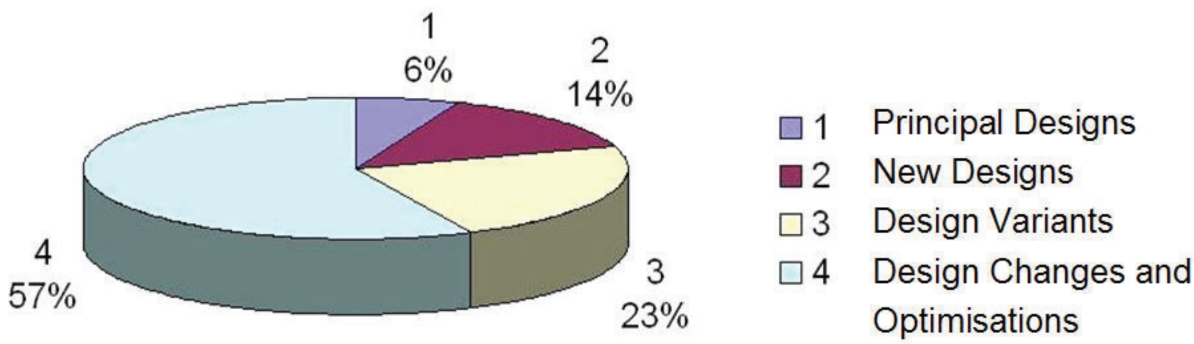


Fig. 4.3 Cost distribution of design costs according to Roller (1997)

In his analysis of four major machine manufacturers with over 1,000 employees using conventional CAD methods Roller (1997) came to the conclusion that 80% of the design costs were spent on designing product variants, modifications and adaptations (see figure 4.3). The high percentage of non-innovative design tasks justifies the implementation of parametric associative design methods, as these tasks benefit in particular from the facilitation of changes and modifications through PAD described in detail in chapter 3 of this thesis.

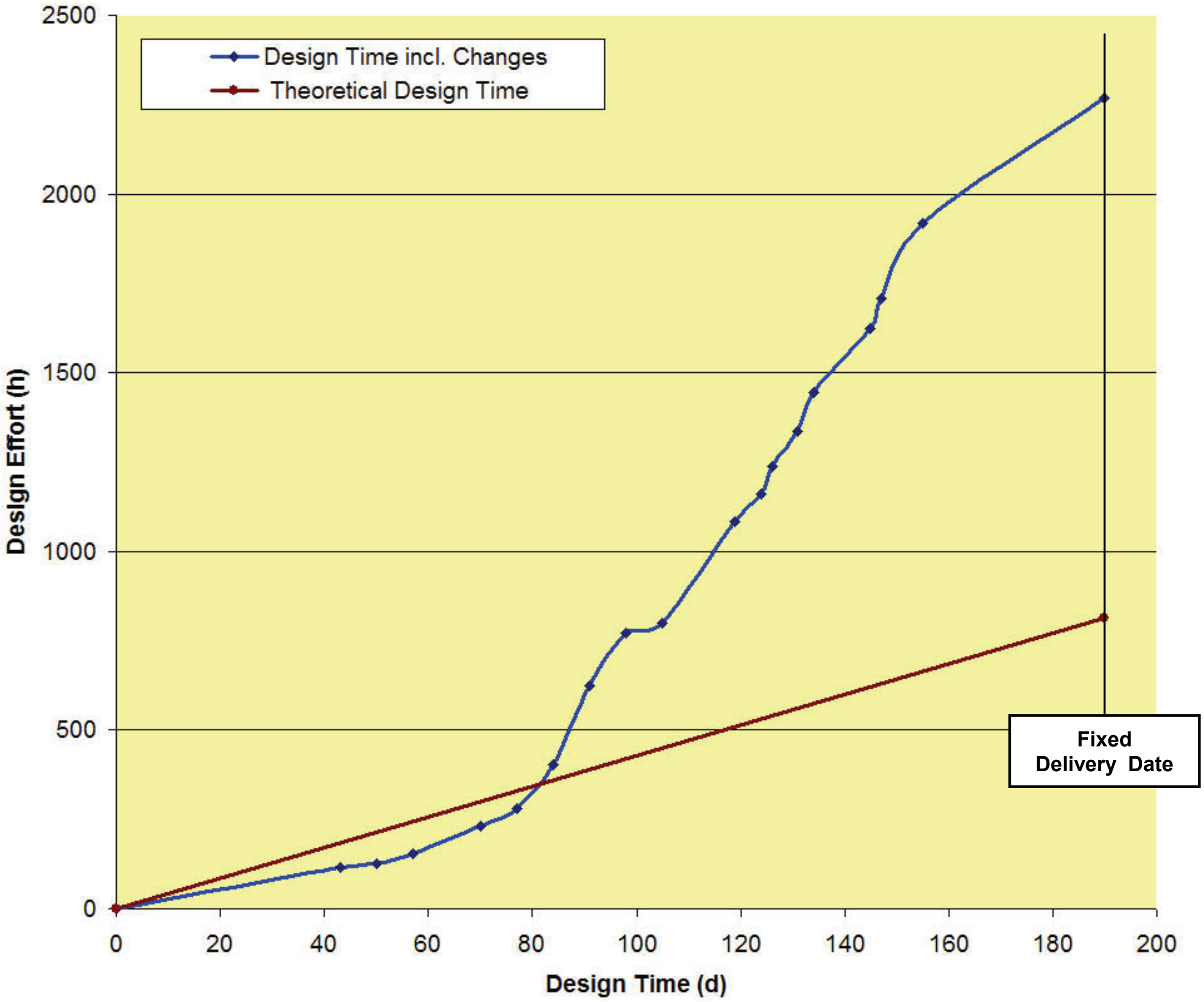


Fig. 4.4 Comparison of planned and actual design efforts (Stüdemann, Schnepp, 2S Engineering)

Figure 4.4 depicts the consequences of unplanned, late changes on the costs of design projects carried out with conventional CAD methods. The scope of the observed project was the design of three cockpit air vents comprising 110 individual parts in total. The diagram shows the initially planned design time (brown curve) and the time actually spent (blue curve). The additional, unplanned time was required for creating a part which had not been considered necessary to begin with, and between two and twelve design modifications per individual part which took between 9 and 40 hours each. A reusable parametric associative design would have caused a higher initial effort in creating the PAD model, however the overall project could have been accomplished with much less effort as the implementation of the changes would not have required the remodelling of the individual parts (Schreiber, 2007; Haslauer, 2006).

As outlined in chapter 3, Parametric Associative Design (PAD) requires a disciplined, well-structured approach to ensure the stability (i.e. robustness) of the models when the geometry is updated after changes and parameter variations have been implemented. When CAD systems with PAD functionality are used, design features are linked to reference elements and their resulting geometry, unless the links are explicitly avoided or eliminated.

The main benefits of a well-structured, traceable modelling procedure are:

- update stability and reusability of PAD models through
- controlled links to reduce the complexity,
- best achievable design quality in each design step, and
- standardised naming of design elements.

Consequently, this chapter will analyse whether available methods for structuring the design and development processes in general can be used to support the methods described above in structuring the parametric associative process chain.

4.1 General systematic Approaches to Development and Design

A design methodology systematises and generalises the design tasks. Whereas the last centuries were dominated by intuitive design procedures, a methodical approach to development and design is favoured today. The advantages of the methodical approach are the improved concentration on the actual requirements of the design output, the avoidance of misleading or redundant development and the reduced effort with respect to time and costs (Ursinus, 2001).

The literature offers several widely accepted standard rules on design methodology (Pahl & Beitz, 2005; Ehrlenspiel, 2006; Hubka, 1992; Roth, 2000; Roth, 2001; Koller, 1994…) which are supported by the VDI 222n family of directives. In this chapter, the

VDI directive 2221 is presented in detail and is compared with the approach suggested by Pahl & Beitz.

4.1.1 VDI Directive 2221

VDI directive 2221 (1993) proposes an industry-independent, universal methodical procedure for developing and designing technical products. This procedure comprises seven individual steps (see figure 4.5) which are performed fully or partially, depending on the specific design requirements, and which may be repeated in several iterations to achieve an optimum solution.

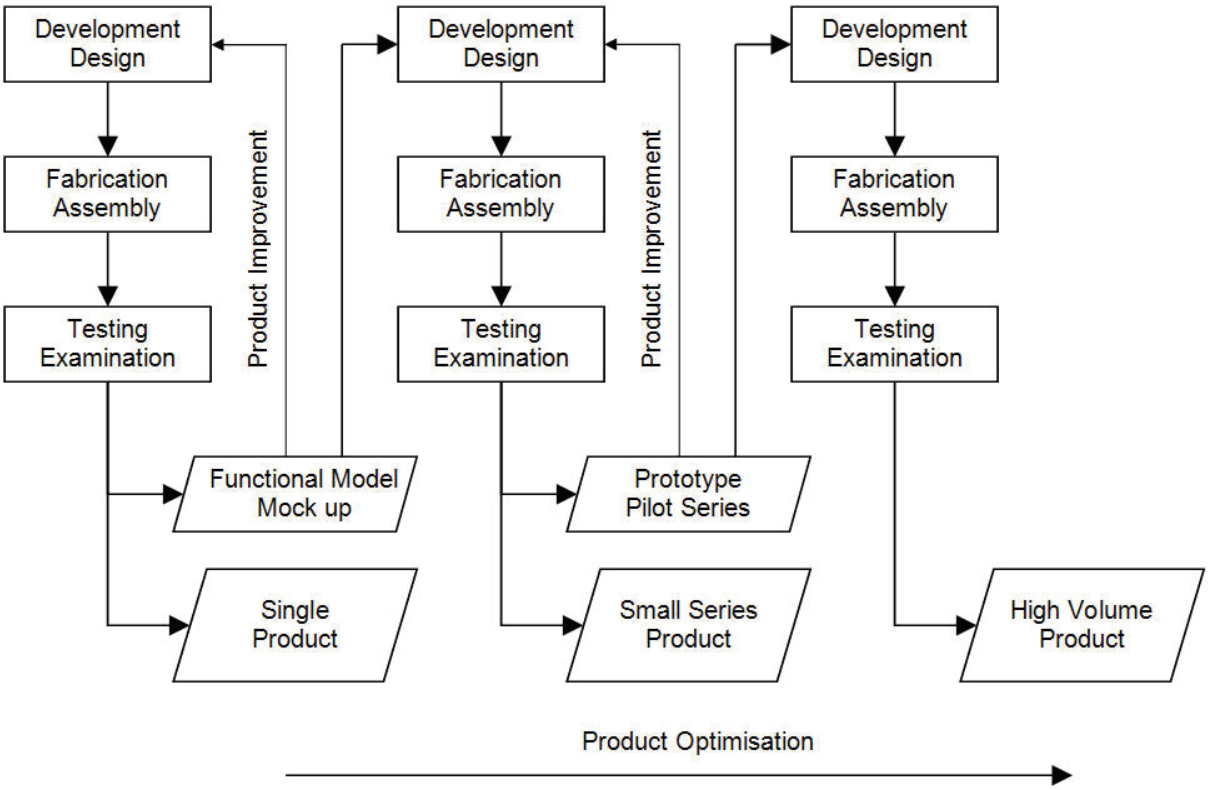


Fig. 4.5 Phases of product development (VDI-Directive 2221, 1993)

Wiendl (2000) has published his concerns that the procedure of VDI directive 2221 may not be a suitable for complex concurrent engineering projects. He doubts that the strict adherence to the VDI directive provides an optimum support for the product development processes: "The VDI directive does not offer a methodical support for managing the timing and the exchange of inputs and outputs between individual parallel and iterative process steps. Stipulating a one-dimensional process flow, the directive does not allow for the flexibility required to manage complex interrelations between the process steps."

The approach suggested in the VDI directive favours the systematic, subsequent arrangement of all development and design processes. It structures the individual steps and defines a standard nomenclature. Permitting the omission and the iterative repetition of individual process steps as well as the going back one or more steps and then moving forward again, the directive reflects the actual design processes. The recommended repetition of individual process steps yields new findings which are valuable input for subsequent process steps, and therefore contribute to the optimization of the developed products.

The directive suggests starting as early as possible with breaking down the development project into small manageable tasks which are suitable for a holistic engineering approach. It thus enhances the early identification of partial problems and restrictions through the definition of structures and interrelations. It proposes the parallel conception of alternative solutions, the reuse of well-known and proven solutions and the introduction of rational organisational job-sharing using concepts like concurrent engineering.

Additionally, the directive suggests a universal approach to problem solving: the process of finding a solution is subdivided into phases which yield more and more detailed results as the project evolves. The directive describes this strategy: "from abstraction or generalization to concrete or detailed solutions". This approach, which is also described as "from interior to exterior" or "from coarse concepts to fine details" will be discussed later in this chapter.

The directive describes the difference between development and design tasks and concludes the following universal procedure for classifying these tasks:

- Was the task requested by internal or external customers?
- Shall a completely new product be developed including the required manufacturing equipment?

- Which degree of novelty is expected of the product – new, variant or adaptive design?
- Shall the product be developed for large batch or one-off production?
- What industry category does the product belong to (e.g. special purpose vehicle, passenger car, aircraft)?
- What are the typical design goals (e.g. light-weight structures, high safety, ergonomic and aesthetic design …)?
- What external restrictions or internal requirements govern the development (e.g. market competition, cost reduction, rapid development, reuse of standard components, design instructions, in-house or external production …)?

The development and design process is of major significance for the whole Product Evolution Process (PEP) and defines production, sales, use and disposal/recycling. Experiences and expectations from previous development projects influence the new PEP. The success of the product development strongly depends on internal communication and the exchange of information with all functions involved in the product life cycle.

The number of iterative cycles required for the successful accomplishment of the development project (see figure 4.5), the extent of the development tasks and the manufacturing aspects must be determined individually for each development project as they depend on the product to be created (e.g. mass product or individual one-off design). Individual designs and small series are usually developed using readily available components and standard parts whereas mass products are generally dominated by special designs tailored to suit the specific requirements of the product. With the growing diversity of mass products, many companies are introducing carry-over and platform concepts to reduce the number of individual parts and increase batch sizes in the production. Generally, the definition of the design details requires much iteration to ensure that all requirements are met and optimum solutions are found for competing and contradictory requirements. It is common practice to define digital prototypes in suitable PDM/CAx systems (Digital Mock-Up = DMU) and verify design decisions through comprehensive calculations and numerical simulations before physical prototypes are built and tested.

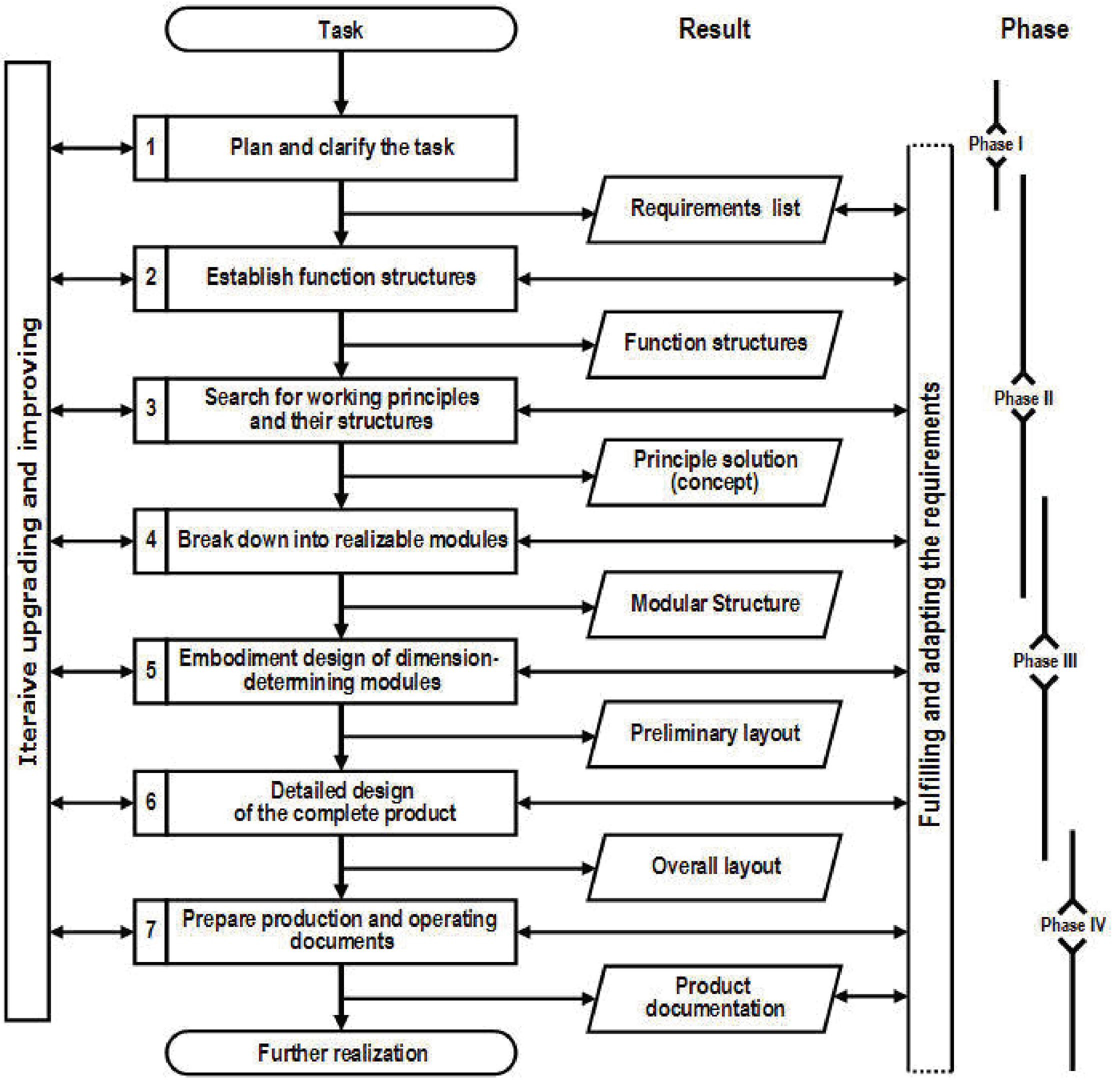


Fig. 4.6 General approach to development and design (VDI-Directive 2221, 1993)

The design process consists of four main phases (see figure 4.6) which the VDI directive simply refers to with the Roman numbers I to IV. They may be called

- planning,
- concept definition,
- embodiment, and
- detailing.

The first of seven process steps (see figure 4.6) is devoted to clarifying and defining a portfolio of design goals, which is later turned into the design specification containing detailed requirements of the design output. These requirements are continuously refined and adjusted with growing knowledge of the specific characteristics of the product and the feasibilities. At a certain milestone within the project, the specifications are frozen and defined as compulsory for all subsequent development steps.

The second process step deals with the definition of the overall function and the associated main and supporting functions of the product or system to be developed. They are documented in a function model, which is an abstract representation of the planned product functionality.

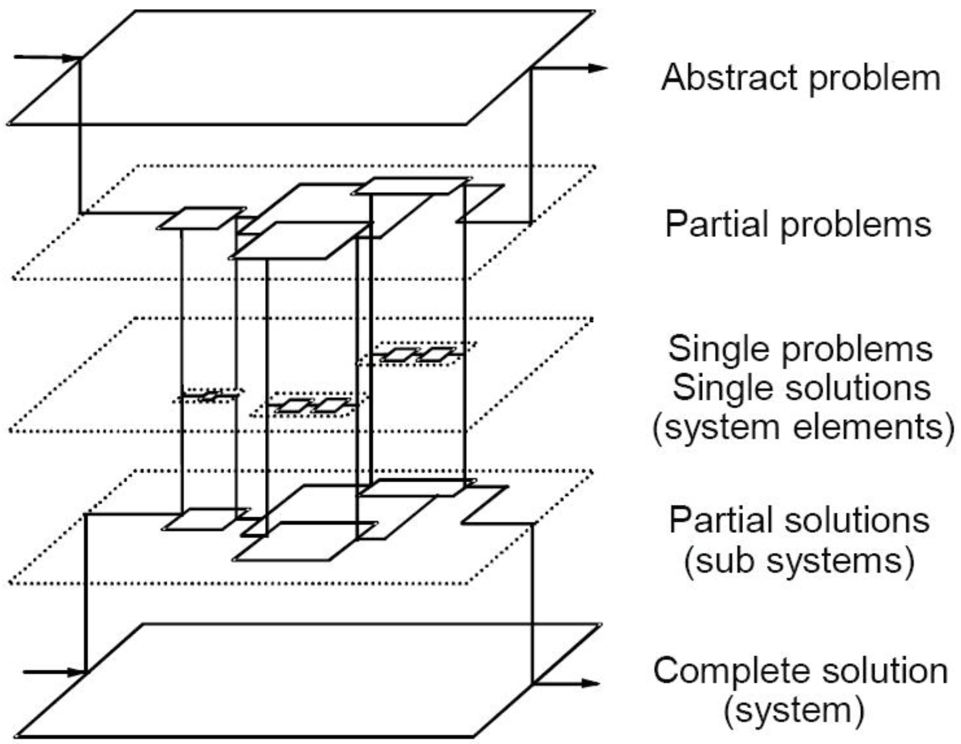


Fig. 4.7 Method of subdivision (encapsulation) and link-up for problem and system structuring (VDI-Directive 2221, 1993)

The decomposition of the overall function into partial functions and their synthesis into feasible structures (see figure 4.7) forms the input of the third process step, the search for suitable solution principles.

The engineering solutions found for individual partial functions are combined into an abstract product model referred to as "effect structure". The output of the third process step is a set of solution principles.

In the fourth process step (see figure 4.6), the solution principles are translated into realizable modules. Contrary to the functional model and the effect structure defined in the previous process steps, the realizable modules already reflect the proposed product structure consisting of sub-assemblies and components (partial systems and system elements) and indicate their interrelations. According to the VDI directive, the modularisation prior to the time-consuming steps of embodiment and detail design is inevitable, particularly for complex products, to facilitate an efficient breakdown structure of design projects and to identify and solve focal points in the development (VDI directive 2221). Even if manufacturing processes and boundaries between individual components have not been finalized, the determination of well-defined interfaces permits the parallel design of variants, individual product zones, sub-assemblies and components which are later synthesized.

In the fifth process step, the variants, individual product zones, sub-assemblies and components are roughly dimensioned. Principal cross-sections and primary surfaces are designed, permitting the optimization of functionality and design. The sixth process step is devoted to the detailed design by adding secondary surfaces and detail features to the primary surfaces and preparing bills of materials and engineering drawings (where applicable). Finally, in the seventh process step, additional documents describing the product, its manufacturing processes and all instructions for use, maintenance and disposal, are prepared. Examples of these documents are production plans, assembly instructions, and user and service manuals.

For computer-aided design processes, the directive recommends the combination of software solutions for the individual process steps to integrated development systems. It is necessary to ensure the continuity of the process chain and the work flow, and the efficient reuse of process step outputs as native inputs to subsequent steps. Integrated development systems permit the flexible application to different tasks and process steps. The core elements of computer-aided development are the product models, which represent the whole product or only parts of it as they contain product-defining data and their interrelations.

According to Liese (2003), the product development process can be considered as a modelling process, in which partial models are created in a sequential performance of embodiment stages comprising the input, structuring and linking of data.

4.1.2 Comparison of VDI Directive 2221 to the Approach of Pahl & Beitz

Pahl & Beitz (2005) recommend a methodical, systematic procedure for engineering design to support the designer in meeting his main responsibility with respect to the technical and economic product characteristics and the large diversity of engineering problems and tasks in the development of technical products. Pahl & Beitz define design methodology as a set of concrete procedures for developing and designing technical systems, which have evolved from the findings and conclusion of design research, psychological investigations, and a large variety of practical experiences. They use flow charts to depict their design methodology and support the practical organisation and management of design projects, and their subdivision into parallel and sequential steps which are interrelated. Additionally, they describe strategies for meeting the design goals and objectives, and methods for solving individual design problems and tasks, to support the designer's intuition with a methodical approach ensuring the successful accomplishment of the design work.

According to Pahl & Beitz, a design methodology shall:

- permit the problem-oriented approach to any design task independent of the associated industry,
- facilitate the identification of optimum solutions,
- ensure reproducible solutions and avoid random solutions,
- support the transfer of solutions to similar tasks,
- be compatible with definitions, methods and knowledge of the corresponding disciplines,
- be suitable for the application of electronic data processing,
- easily taught and learnt,
- help to facilitate design work, save time, avoid false decisions and ensure active and devoted cooperation of all persons involved,
- facilitate the planning and control of project work in an integrated, multi-disciplinary product realization process, and

- serve as a guideline for team and project leaders.

When comparing the VDI directive 2221 with the methodology published by Pahl & Beitz, it becomes evident that both approaches divide the product definition process into the following four main phases: planning and clarifying the task, conceptual design, embodiment and detail design. Both approaches stress the need for multiple iterations in individual process steps to optimize product functionality and topology.

VDI 2221: Working Stages and Results
1. Clarify and define the task Specification
2. Determine functions and their structure Funktion structures
3. Search for solution principles and their combinations Principle solutions
4. Devide into realisable modules Module structures
5. Develop layout of key modules Priliminary layouts
6. Complete overall layout Definite layouts
7. Prepare production and operating instructions Product documents

Pahl & Beitz: Working Steps and Results
1. Plan and clarify the task Requirements list
2. Develop the principle solution Principle solution (concept)
3. Develop the construction structure Priliminary layout
4. Define the construction structure Definitive layout
5. Prepare production and operating documents Product documentation

Fig. 4.8 Comparison of VDI directive 2221 and Pahl & Beitz

While the general approach of the VDI directive 2221 comprises seven process steps, the methodology of Pahl & Beitz defines only five process steps. The difference between both step models is that the principle solution stage is considered as one process step by Pahl & Beitz whereas the VDI directive subdivides this step into two steps (Step 2) and (Step 3) and the process step of structure and preliminary layout by Pahl & Beitz is subdivided into two steps (Step 4) and (Step 5) by VDI directive (see figure 4.8).

Both approaches break down the embodiment stage, which aims at the definition of the material, geometric and manufacturing characteristics of the product, into four

Process steps. Despite the evident differences in the nomenclature and the degree of abstraction of these approaches, they can both be considered as representative of today's procedures in the development of automobiles and their components.

According to Pahl & Beitz, only 3D CAD systems are capable of supporting the integrated computer-aided Product Evolution Process (PEP) with a product model derived from a unique data base.

Liese (2003) states that the descriptions of design processes, which can be found in the VDI directive 2221, and the works of Pahl & Beitz and other renowned authors are always formulated as work instructions. "To gain a complete understanding of the product development process, not only the required process steps must be known but also their interrelations, their timing restrictions and the required information flows" (Liese, 2003).

4.2 Detailed partial Methods of structured 3D CAD Modelling

4.2.1 Strategies of increasing Concretion

Most publications on problem solving share the conclusion that solution finding processes should be performed in a sequence of cycles with increasing concreteness. A selection of these strategies shall be explained here.

4.2.1.1 Strategy "from coarse Concepts to fine Details“

The geometry is determined in one or several initially separated design zones, and the created CAD model is refined in several steps with increasing detail, evolving from a coarse concept to a detailed, fine design. The work on individual design zones can be accomplished independently, with different degrees of maturity. The design zones can be assembled to larger zones or complete products by joining, trimming, filleting and other operations. The detailed design is derived directly from the coarse topology which was defined in the early design stages.

This strategy is the dominating method for designing car body panels, where the design process starts with primary surfaces and is refined through the generation of secondary surfaces, geometric details (features), fillets etc.

4.2.1.2 Strategy "from Exterior to Interior"

When the created products require an aesthetic exterior, the overall styling of all visible components is accomplished before the engineering design task is started, and broken down into individual problems/tasks. This strategy applies to car body design as well as the design of consumer products. The overall visual impression of the product is the main driver dominating the engineering design process, and functional aspects may be considered subordinate to a certain extent (VDI directive 2221).

The predominantly styling-oriented design process of a car body starts off with the free-style shaping of the exterior appearance. Once the initial draft of the vehicle (general layout and package) is prepared, the styling is brought in line with this draft. The resulting visible surfaces are then analysed to ensure that styling, functionality and manufacturing requirements are met.

On the basis of these exterior styling surfaces and characteristic curves, the functional parts and sub-assemblies of the car body are designed, starting with the exterior body panels and ending with interior parts like reinforcement panels etc.

4.2.1.3 Strategy "from Interior to Exterior"

The design of a machine (e.g. an automotive engine) is primarily dominated by the desired function. The exterior appearance plays a very minor role and it merely governed by aspects of manufacturability and safety, and the interfaces with other components.
Together with mechanical strength and passive safety requirements, the space occupied by the automotive engine and its accompanying components as well as the

front axle (and the front wheel suspensions) have a strong influence on the exterior shape.

The vehicle interior is designed on the basis of passenger ergonomics (seating package), taking into account the space the passengers need to feel comfortable, their visual and haptic sensations, as well as active and passive safety requirements. While the exterior shape of a passenger car is designed with the strategy "from exterior to interior", the car interior and the surrounding window surfaces are designed from interior to exterior according to the needs of passengers and users.

4.2.2 Modelling Principles

In every development project, engineers should strive to model their designs with an optimum robustness (update stability), and create comprehensible and clearly structured geometric elements and specification trees. Another important requirement is the suitability of the design task to be accomplished by several co-workers in collaboration. To satisfy all these constraints, several modelling principles have been developed, which will be described in this section.

4.2.2.1 "Bottom-up" Principle

When all individual parts are designed independently, and subsequently joined together to sub-assemblies and complete product structures with the aid of assembly constraints, the modelling procedure is called bottom-up principle. Only in the final stage of this modeling process, when the parts are assembled, can the designer verify that all parts fit together without collisions, and that the overall assembly meets the requirements with respect to functions and shape. All control parameters and references are defined within the separately modelled individual parts. There are no links between the individual part models.

The systematic assembly of the separately designed components can be facilitated by simple positioning geometry (points, straight lines, planes…) which are used to establish the assembly constraints between the parts when joining them together in a product structure.

Typical parts which are suitable for the bottom-up principle in automotive body design are standard parts, standardized components, and carry-over-parts. In the design of small batch and one-off products, this principle is also used. In the design of mass products like automobiles, the mixture of repeated and unique (new) parts leads to the combination of bottom-up and top-down design.

4.2.2.2 "Top-down" Principle

On the basis of a previous design, the empty product structure of the new sub-assembly is defined. The product structure identifies the required sub-assemblies and individual parts of the product. In the automotive development and production planning processes, engineering bills of materials (EBOM) and manufacturing bills of materials (MBOM) are defined. In addition to the actual parts and sub-assemblies, the engineering bill of materials contains additional engineering models, like styling surfaces, package models or kinematic simulations, which are not contained on the manufacturing bills of materials. When the MBOM is derived from the EBOM, manufacturing aids and media are added, like jigs and fixtures or lubricants and adhesives.

Once the responsibilities for the design of all sub-assemblies have been assigned and the driving parameters and reference geometries as well as anticipated modification scenarios have been defined, the empty start-up models of the parts and sub-assemblies are filled with suitable geometric elements or existing geometries of previous designs which are adapted to the current design task.

Modeling is performed in the context of the sub-assembly. The Individual parts are controlled by reference geometry and parameters on the sub-assembly level - these elements can only be modified in the sub-assembly and do not belong to individual parts. In some cases, the geometry of a part may be based on parameters and/or reference geometry from another part which belongs to the same sub-assembly.

4.2.2.3 Automatic Link and View Link

The "automatic link" principle defines links between controlling parameters or reference elements and dependant geometry in a dynamic way, so that the modification or the replacement of the control elements causes the automatic adaptation of the dependent geometry according to the engineer's design intent.

Derived from the "automatic link", the "view link" principle contains all links between driving reference elements or parameters and dependent (driven) geometry twice:

a) Isolated links are used to design the driven sub-assemblies and individual parts, to avoid their automatic modification when driving reference elements or parameters are changed. The designer can take his time to assess the consequences of the changes on his model.

b) Parallel to the isolated links, so-called "view links" are established which are dynamically linked to the reference elements or parameters. These links show the designer that changes have occurred in the reference elements without automatically triggering any modifications in his model.

c) If the designer decides to accept the changes of the driving elements announced in the "view links", he must replace the isolated links in his model with new isolated copies of the changed automatic links.

In chapter 3.4.4, these principles are described in more detail and illustrated in a figure.

4.2.2.4 Lean Hierarchy

To keep the network of links resulting from "parent/children" relations of a PAD model as comprehensible and simple as possible, the hierarchy of elements must be kept as lean as possible.

Brill (2006) recommends not exceeding five hierarchical layers in the specification tree. Hierarchical structures are obtained by combining sub-structures of identical significance to main structures and storing all reference elements and parameters which govern these sub-structures at the head of the main structure. Cross links between sub-structures of identical significance should be avoided.

The complexity of the links within a product model can be further reduced by using symmetry, copy or pattern functions of the CAD systems (Liese, 2003).

4.2.2.5 Zone-based Modelling

The subdivision of a larger sub-assembly into design zones permits the parallel (or simultaneous) accomplishment of design tasks by several designers. In independent investigations conducted by Opel, and the HAW Hamburg in cooperation with Volkswagen Nutzfahrzeuge this approach of concurrent engineering was verified as recommendable for the concept design phase.

This principle permits the separate storage of the geometry pertaining to individual design zones (e.g. a B-pillar with cross-sections to the body panels and the front and rear doors) for later reuse and combination with alternative design zones in a new automobile project or an automobile variant.

The designers who use this approach for a collaborative design project must refer their geometry to control elements of a main adapter. For all interfaces between the zones, specific interface adapters must be defined, and the neighbouring geometry must be referred to the interface adapter.

4.2.2.6 Transparent Modelling

To make a 3D CAD model as "transparent" (i.e. understandable) as possible, it is recommended to use standardized names for all objects in the specification tree of the model. The names should reflect the function and the context of the elements (Liese, 2003).

Many design tasks require the same design logic of the geometrical sub-sets, leading to identical patterns of geometrical elements. If this design logic is standardized, the patterns can be easily reused within the same design. Together with standardized naming conventions, the repetition of design patterns enhances the comprehensibility of CAD models to people who where not directly involved in the creation of the model. Misinterpretations and misconceptions can be avoided and designers find it much easier to modify and adapt models they have not created themselves (Brill, 2006; Liese, 2003).

4.2.2.7 Model Parameter Variation

The desire to establish robust PAD models which can be reused requires the foresighted structured use of control parameters and references. The interior consistency of the models must be ensured by the definition of parameter sets, ranges and families, and their possible variations must be checked through iterative variations and validated before the CAD model is released.

4.3 Object-oriented Design

Object-oriented programming has been used in Europe since the mid 80s and has influenced CAD modelling as this programming technique is now state of the art for modern 3D CAD systems. The fundamentals of object-orientation are also applicable to the parametric associative design process, which is then referred to as object-oriented design. While the user of conventional CAD systems concentrates on generating explicit geometrical elements, PAD models are created with the focus on objects and their functionality.

Objects are tangible and visible things which fulfil a certain purpose. As part of a product to be developed, all objects (should) contribute to the main function of the product, either directly or indirectly. According to the required partial functions of the design, the object-oriented design engineer defines individual design zones which can be considered as objects of a mechanical part or a sub-assembly. Examples of such objects are e.g. the recess of the door inner panel for the door latch or the recesses in the B-pillar for the door hinges. As the function and the design of both "recesses" are similar, a class "recesses" can be derived and defined. Object classes contain basic definitions of similar objects, i.e. each object belonging to the same class inherits the attributes and the behaviour from its class.

The advantages of object-oriented design are the same as those which have led to the success of object-oriented programming:

- reuse of designs, and
- ability to control complex model structures.

4.3.1 The Object-oriented Design Process

According to Brill (2006), Schäfer (1994) and Booch (1994), the object-oriented design process can be divided into the phases (see figure 4.9):

- conception,
- analysis,
- design,
- evolution, and
- maintenance.

4.3.1.1 Conception

Wikipedia (2008:2) defines the conception (n; verb: conceive, adjective: conceptional, from the Latin concipere: to take in, from capere to take) as a comprehensive collection of objectives and derived strategies and measures for the implementation of a larger idea requiring strategic planning. The concept contains the necessary information and reasoning, often accompanied by a risk analysis, an action plan and a schedule. In the automotive development projects, the results of the concept phase are documented in a requirement specification.

4.3.1.2 Analysis

In the subsequent analysis phase, the objects and classes of the design zones to be designed are identified and their interrelations and dependencies are determined.

For graphic representation of these relations and the necessary discussion, mind maps are a useful tool. Due to the high complexity of interrelations and dependencies, the initial concept mind map often lacks comprehensibility. This disadvantage can be avoided by hierarchical mind map structures (see figure 4.10), which can serve as a valuable basis for defining a top-down structure of the design and the associated sub-assembly hierarchies.

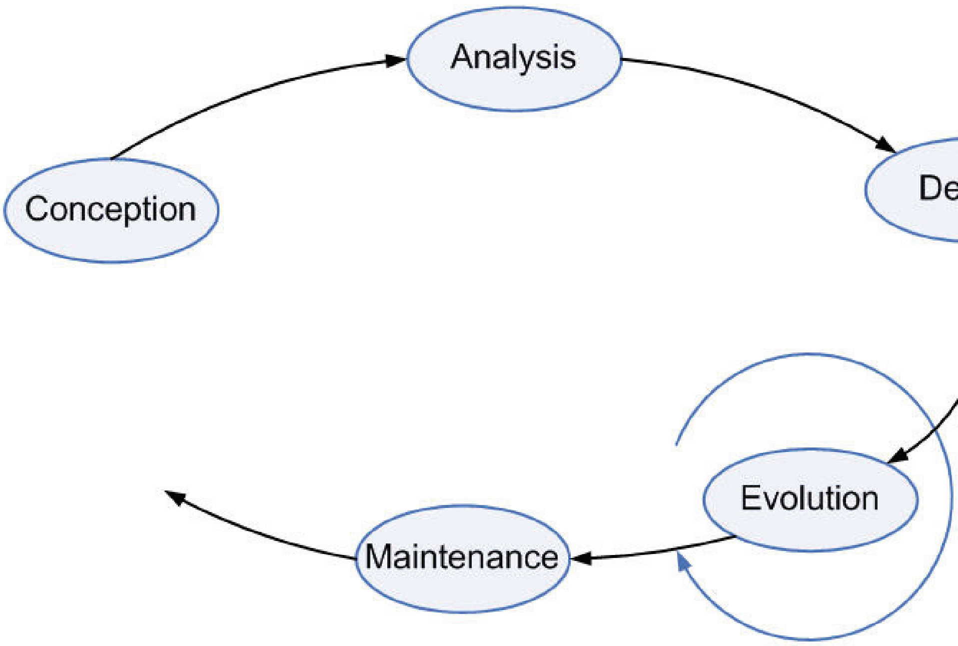


Fig. 4.9 The object-oriented design process according to Booch (1994)

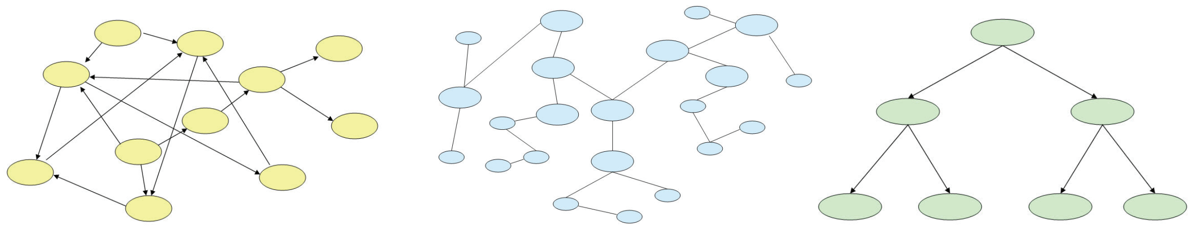


Fig. 4.10 Mind maps – from left to right: concept mind map, linked and hierarchical mind maps

4.3.1.3 Design

In the design phase the initial geometry is created in the CAD system.

4.3.1.4 Evolution

The continuous optimization of the sub-assembly to be designed is performed in numerous iterations on the process chain from concept development to final design. Each iteration is a challenge to the stability and robustness of the CAD model which should be improved continuously, based on lessons learned.

4.3.1.5 Maintenance

Once a CAD model is released, it must be maintained. This stage does not differ much from the evolution phase. Additional refinements and the addition of new functions increase the robustness of the model and open up the reusability of the model in other project contexts.

4.3.2 Important Principles of Object-Orientation

The principles of object-orientation shall help to control the complexity of a task. In the works on object-oriented programming, we find different classifications of the basic elements of object-orientation. The main elements of object-orientation as described by Booch (1994) are:

- abstraction,
- encapsulation,
- creating hierarchies, and
- modularisation.

4.3.2.1 Abstraction

Abstraction denotes the concentration on the essentials. To abstract means to omit details and to eliminate references to specific examples through generalization and simplification. With the aid of such a "fuzzification", objects can be classified, and object classes with common attributes can be defined.

In the context of automotive body design, abstraction means for example to identify and design of the essential primary surfaces of a part, and to postpone the design of less important secondary surfaces and details to later design steps. Brill (2006) states the definition of the class “reinforcing bead” containing the objects “roundly bead” and “trapezium bead” as another example of abstraction. Both objects have many identical design details.

4.3.2.2 Encapsulation

Encapsulation means the isolation of a partial problem or system from its surroundings, and the reduction of the relations to the surroundings to an interface. According to Schäfer (1994), encapsulation and abstraction complement one another. While abstraction focuses on the exterior aspects of the objects, encapsulation concentrates on the internal aspects. Encapsulated partial systems or objects with identical interfaces can be replaced independent of their internal structure, and they can be reused without limitations within the same model or in other CAD models.

"All components of an object are encapsulated, i.e. isolated from its surrounding, which is replaced by defined interfaces. From the overall system perspective, the object appears as a black box. The complexity is reduced either by the concentration on the exterior characteristics and the integration into the overall structure or by the concentration on the internal design with all its numerous details and the negligence of the overall structure (Brill, 2006)."

4.3.2.3 Hierarchy

Wikipedia (2008:3) defines hierarchy (from Greek ιεραρχία = hierarkhia) as combination of ιερή, hieré = holy and αρχή, "arché" - control, order, principle, the beginning, the first. The term is applied to a system of objects arranged in a graded order. It denotes a series of ordered groupings within a system, such as the arrangement of plants and animals into classes, orders, families, etc. Consequently, the classification of objects in a hierarchical system implies a ranking of these objects.

The manufacturing bill of materials defines the hierarchy of an assembly with respect to its sub-assemblies and parts. Starting off from the top node (the root), the hierarchical tree branches split off into subordinate assemblies and individual parts.
The order in which parts are designed and the decisions on their reuse determine the design hierarchy of the parts or design zones.

4.3.2.4 Modularisation

The term module denotes a standard self-contained unit or item that can be used in combination with others. Products or systems with a modular design therefore consist of an assembly of standardised modules.

Brill (2006) defines modularisation as the process of determining which sections of the object hierarchies shall be saved in individual files. He considers abstraction, encapsulation and the definition of hierarchies as the logical internal structures, and modularisation as the physical structuring of the data. As examples of design modularisation, Brill states the decomposition into separate files like external references, concept geometry, product geometry creation, drawing generation, and associated process documentation.

4.4 Summary

All of the above methods and process steps have in common that several solution variants are investigated, calculated, simulated, built as samples or prototypes and tested, to continuously improve and optimise the output of the development process.

Investigations of the author have shown that the development of mass produced automotive body parts needs several iterations to optimise parts and assemblies according to different, in parts conflicting goals throughout the different phases of product development.

PAD is a kind of object oriented design. The design approach and definition of the structural tree require the consideration of all methods and strategies explained in this chapter to guarantee the update safe exchange of reference geometry and parameter variations as well as the legibility and the reusability of PAD models. Nevertheless it is the task of the OEMs to settle mandatory rules for all engineers involved in the development process. The German association of the automotive industry (VDA) has a centre (VDA-QMC) which defines rules for the cooperation between OEM and suppliers to improve quality and standards. Besides this centre there is a user group of all German OEMs using CATIA which defines common rules for the product development process.

In the product and production process development of mass products like automotive bodies and their components, the design and development tasks are distributed among numerous engineers within the OEMs and their subcontractors. All of them must meet the same overall product and project requirements. The single basis of most decisions, validations and verifications, as well as new developments, variations and improvements are the 3D design data sets. Consequently, the next chapter will discuss in detail the process chain of automotive body development and the distribution of tasks within this chain.

Chapter 5. Set-up and Activities of the Product Evolution Process (PEP)

For the development of a new passenger car which shall be produced in large batches, comprehensive and careful planning is required to meet the numerous conflicting design and manufacturing constraints. All relevant initial strategic plans are developed by a product strategy committee. The overall responsibility is assumed by a member of the chief executive committee of the OEM or by a management representative appointed for the project. Once the product strategy committee has decided to launch a new automobile project, the Product Evolution Process (PEP) is prepared, comprising product planning, product development and production planning. For this purpose, the OEM has defined master processes permitting the detailed planning of the project and the reliable validation of the maturity levels. To support this, the Quality Management Centre (QMC) of the German Association of Automobile Manufacturers is currently trying to establish a nation-wide process of maturity level validation between OEM and Suppliers (VDA, 2006 & Klein, 2006 & Weißbrich, 2007). The nature of the development project (e.g. a facelift or a completely new passenger car) determines the duration and the details of the development and planning scope of the required product development processes.

The master processes describe and control the different maturity processes in the product evolution, e.g. the maturity processes for the automobile package, the styling, the module definitions, the prototype phase or the mass production phase based on the analysis of several independent publications (Braess & Seiffert, 2007:1 & Damme, 2003 & Dietze, 2001 & Form, 2006 & Gevert, 2003 & Klein, 2006), the master processes of three German OEMs (BMW, Daimler and Volkswagen) have been reconstructed and compared (Fig. 5.1). According to the literature, the master process of a conventional automobile development has duration of approximately 54 to 60 months. The project timing is structured with the aid of defined phases and milestones (Volkswagen), Quality Gates (Daimler) or synchronisation points (BMW). The milestones are defined control points (often documented in project reports) at which certain results or findings must be available and a certain product maturity level must be achieved. It is the objective of the master processes to define the maturity level validation (to ensure the reliability) in the product evolution (PEP). Phases and milestones are defined for each new project before the project is kicked off.

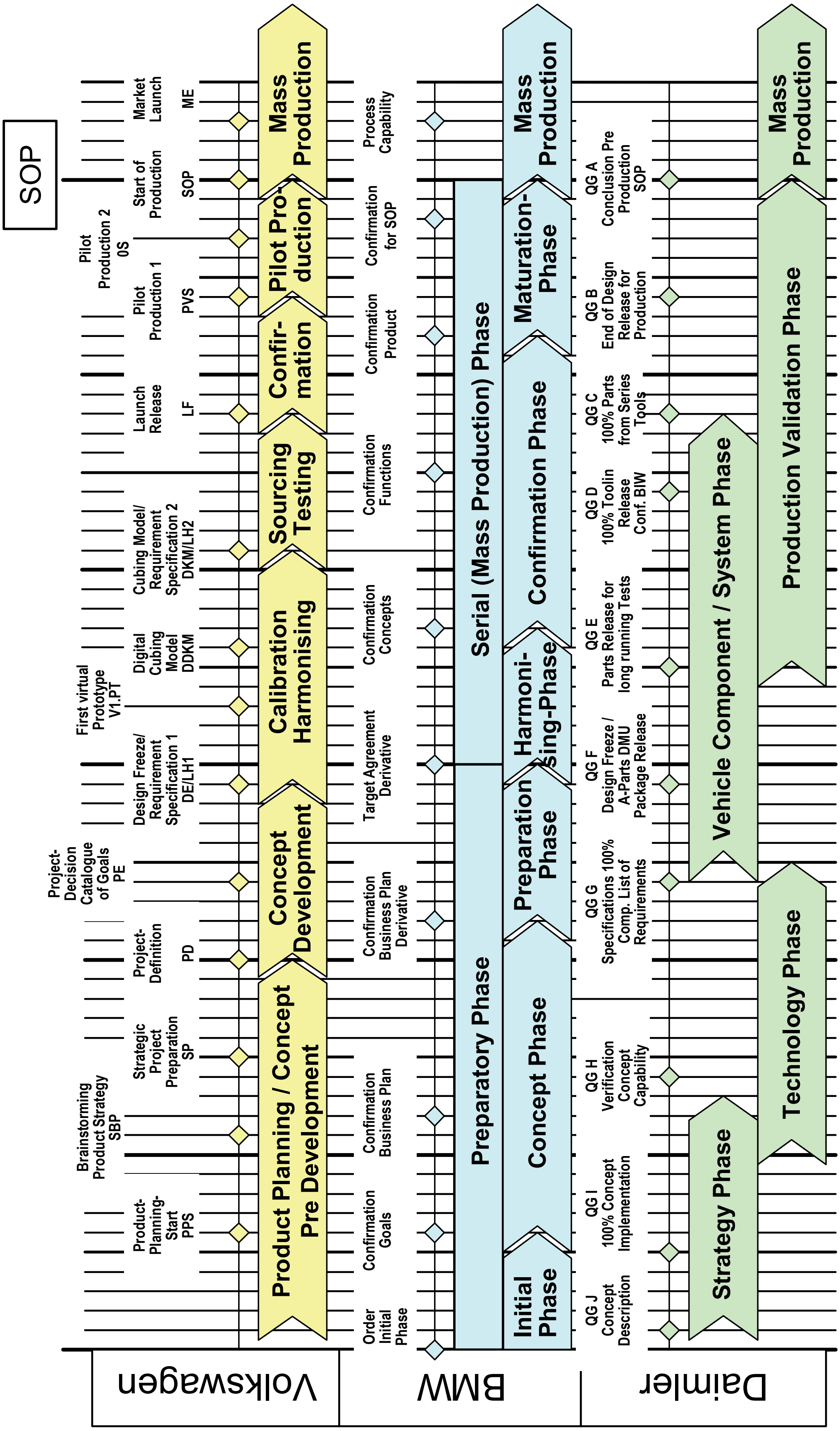


Fig. 5.1 Master Product Evolution Processes (PEP) acc. to Braess & Seiffert (2007:1), Damme (2003), Dietze (2001), Form (2006), Gevert (2003), Klein (2006)

Independent of a specific automobile project, the OEM and system suppliers continuously monitor the market. This often leads to general investigation and research projects which are yet independent of specific car or system developments. As described in chapter 4 (see figure 4.6), the PEP comprises analyses, definitions, coarse and detailed design steps, which are iteratively performed, often jumping back and forth within the steps, and finally leading to optimized solutions.

The development of a new automobile is generally performed independent of the aggregate development. Several other modules and components like front seats or axles and suspension systems are rarely developed with reference to a specific car project, i.e. they remain unchanged in their basic design and are only adapted to the special requirements of the car project.

Nowadays, most OEMs have adopted platform strategies for deriving different models and production lines from a standardized design basis. Generally, a platform is defined as the reusable design of the floor pan components, which is only of indirect influence on the styling and design of the visible parts of the body-in-white but has uniform reference points for the automotive body production (clamping and positioning points) and for attaching power train components like engine, transmission, and wheel and suspension system etc.. The sophisticated process of defining platform concepts is based on comprehensive analyses and research projects carried out by all OEMs. The consequence of the platform strategy is that the early phases of car body design can and must use a set of carry-over parts which are available as 3D CAD models. The selection of the correct platform components for the product family must occur at the beginning of the design project and must be verified and validated as soon as possible with the highest achievable reliability. BMW subdivide their master PEP which follows the strategic product planning phase, into the following two major phases: “preparatory phase” and “serial (detailing) phase”, each of which are allowed to take approximately 30 months. Both major phases are split up into three subordinate phases. The preparation phase is broken down into “initialization phase”, “concept phase” and “design phase”, and the detailing phase is broken down into “agreement phase”, “confirmation phase” and “maturity phase” (see figure 5.1).

The objectives of the first major phase, the preparatory phase, are (see figure 5.2):

- investigate, define, agree and verify design, technology, and innovations of the basic model of the new car family,
- investigate, define, agree and verify design, technology, and innovations of the cars to be derived from the basic model (this process begins after the basic model has been defined),
- determine timing and responsibilities for the serial phase,
- eliminate initial conflicts between design objectives as far as possible,
- accomplish and freeze package and styling,
- derive the engineering bill of materials, and
- elect development partners.

This phase is responsible for defining about 75% of the product costs, and it has a major impact on the company's profit of the next decade.

In the process of converging package, styling and engineering, which is a part of the preparatory phase, the approach is from interior to exterior (with the focus on technical constraints, i.e. "form following function“) as well as from exterior to interior (with the focus on styling, i.e. "form following emotions") (Braess, Seiffert, 2007:2). The design freedom of the car body is restricted by the styling-oriented exterior shape of the car and the function-driven package. This leads to numerous conflicting and contradictory objectives which must be resolved in multi-disciplinary convergence processes.

The objectives of the second major phase, the serial (mass production, detailing) phase, are listed as follows:

- The harmonization of the development output (i.e. the results of the preparatory phase) with current legislative requirements, standards, competitive situations and research conclusions,
- the detailed design of individual parts and zones, and their optimization and verification in the context of the complete car design through calculation, simulation, testing and prototype investigation,
- the accomplishment of the homologation legislation and type approval, and

- the planning and production of series production tools and fixtures, and their maturation through sample inspections and pilot production to ensure the capability of all production processes.

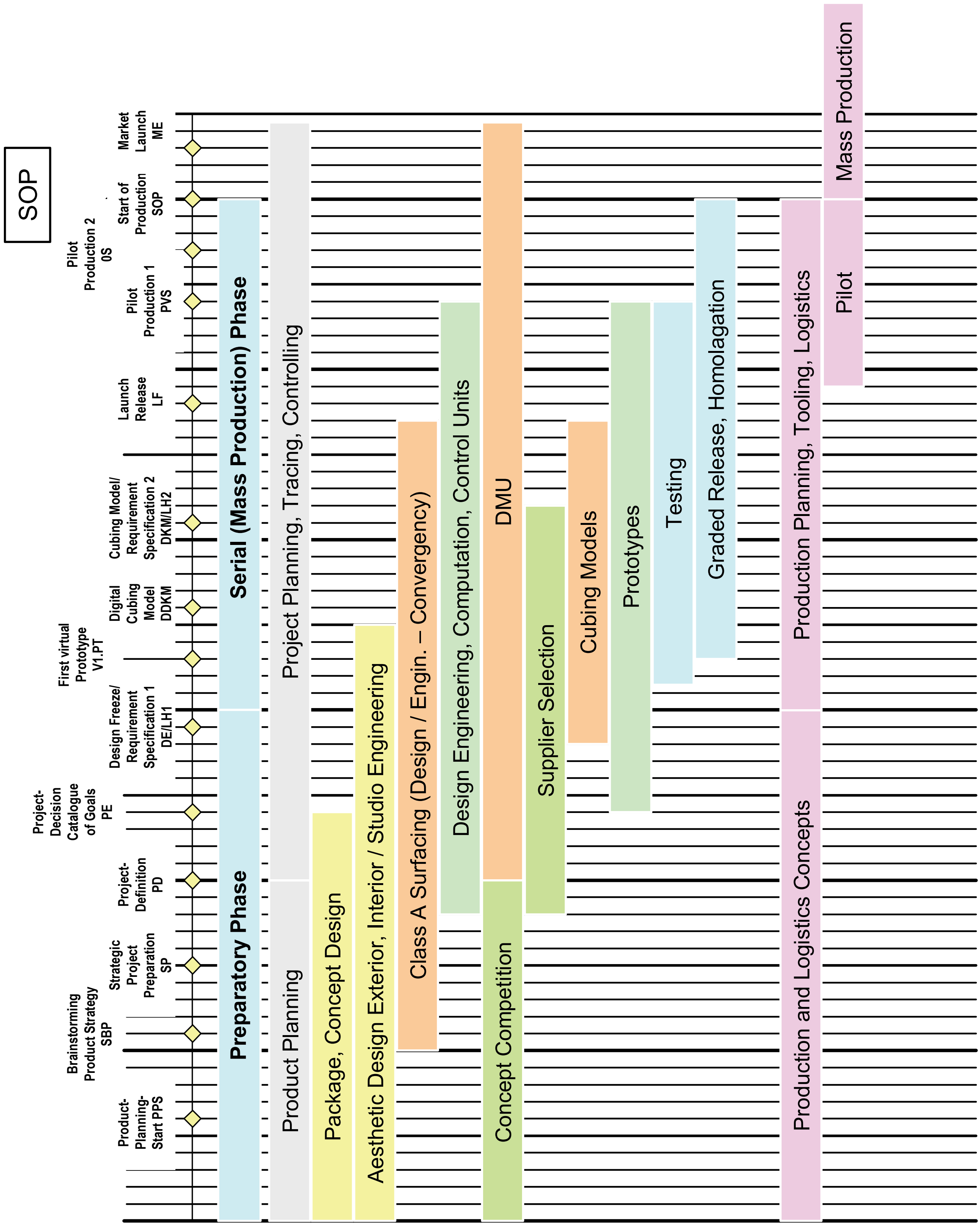


Fig. 5.2 Distributed tasks in PEP acc. to (Dietze, 2001 & Form, 2006)

In the following section, the most significant steps of this second major phase in the evolvement of a new passenger car are described and analyzed from the designer's point of view.

5.1 Packaging / Ergonomics Process and Concept Development

“The packaging process manages and harmonizes the requirements of component locations, ergonomics, and the overall characteristics of a car. It is a multi-disciplinary process which accompanies the complete product evolvement cycle, from the first ideas to the end of production. The administration of all geometric data of the car and the control of all associated documents (to safeguard that each document is up to date) also belongs to the responsibilities of the packaging process. “(Grabner & Nothhaft, 2006)

It is the objective of the packaging process to control and harmonise the interior of the passenger compartment (e.g. the arrangement of the seats) and luggage compartment, the location of power train components, and resulting characteristics like wheel base, front and tail section and ground clearance. The basic car concept defined at the beginning of the development phase is continuously substantiated as the development progresses and package plans, styling concepts and subassembly designs are getting more and more detailed and refined. In the past, this process of converging package, styling and design was a two-dimensional process. Today, the availability of digital 3D tools (3D CAD, Computer Aided Styling System (CAS), large scale projections, power wall, Virtual Reality (VR) caves, 5-axis milling machines, rapid prototyping etc.) has lead to a dramatic acceleration and improvement of the development process.

In the first phase of the packaging process, about 60 to 54 months before Start Of Production (SOP), initial objectives are defined on the basis of strategic specifications derived from market analyses and predictions as well as project-independent results of technical investigations and research. The resulting concept package is a first 3D package, which initially comprises only about fifteen components (Gessner, 2001) of power train, wheel and suspension system (including their interfaces with the car body), the locations of passenger seats and the driver's fields of vision and the envelope of luggage compartment. It is supplemented with components from previous car projects, permitting the rapid qualitative visualisation of the new car concept. In addition, this concept package documents the resulting basic car dimensions like overall length, height, width and the location of the engine and the axles, as well as the anticipated positioning of the driver and the other passengers. On this basis, a dimensional design concept is derived which contains a preliminary coarse of hard points. In this phase between five and six competing concepts are developed in parallel, e.g. with varying wheel base. Additionally, pre-development issues are defined, which must be tackled and solved to ensure the realisation of the package concepts.

Once the initial design objectives have been determined, the second phase (about 54 to 40 months before (SOP) is devoted to the preparation of a correct and marketable overall car concept. All major open issues contained in the initial concept are resolved to obtain a first draft concept containing already about 50 sub-assemblies of the new car. Conflicts between styling and package are resolved by compromises (convergence), leading to an agreed plan of hard points.

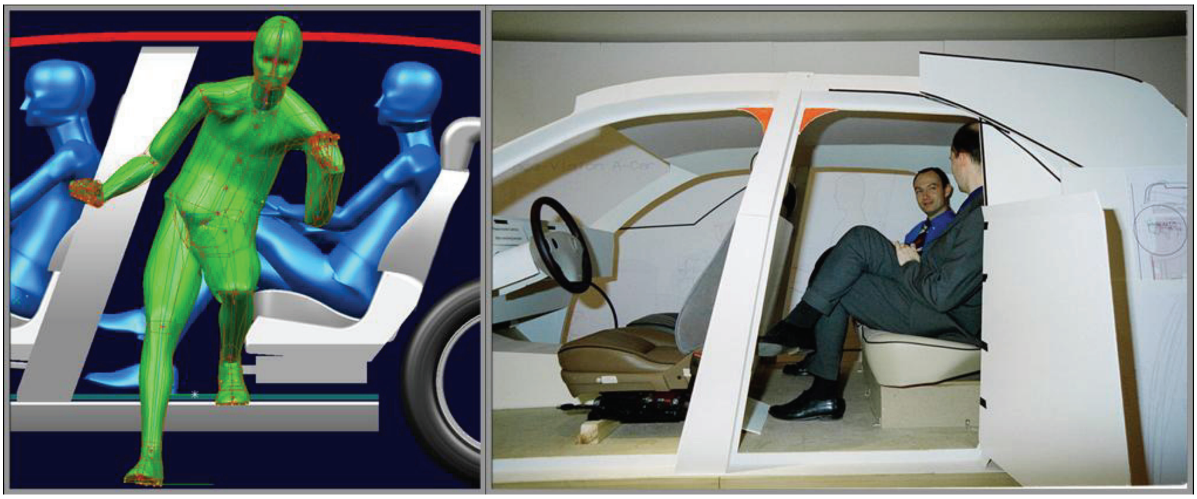


Fig. 5.3 Validation of ergonomics with RAMSIS and seating mock-up on Rolls-Royce Phantom (Lindermaier, 2006)

Furthermore, the engineering concept of the automobile project is determined in this second phase, to permit the verification of the package of wheel and suspension system and power train. With the aid of up-to-date manikins (virtual anatomical models of the human body) ergonomic investigations are made to validate seating positions, viewing angles and access to the car (see figure 5.3). The virtual analyses are verified with a first physical seating mock-up.

Based on the initial design objective, the automotive body development process prepares a concept for the automotive body structure. The overall car concept, which is prepared in the concept phase, is aligned with these first automotive body analyses. Initial Finite Element Method (FEM) calculations are made to verify the overall car concept with respect to structural strength and stiffness. Additionally, concept analyses are performed in all domains of the automotive body development, to harmonise the overall car concept with the structural concept. If the initial design objectives are proved to be feasible, the second development phase leads to the confirmation of these objectives. A final plan of hard points and the first vehicle integration plan are the output of this phase.

The vehicle integration plan (see figure 5.4), which is documented in engineering drawings (scale 1:1), tables and written descriptions, contains all major interior components like seats, steering wheel and other cockpit elements, as well as the Society of Automotive and Aeronautical Engineers (SAE) manikins and Seating Reference Points (SgRP). The space occupied by power train and wheel and suspension system is visualized in the three main views. Furthermore, all major dimensions, driver viewing angles, nominal loads, slope angles etc. are documented in this plan. In the past, the resulting two-dimensional geometry was the basis of the styling tape plans. These tape plans (scale 1:1) were used to optimise styling geometries to and initiate the 3D design process.

Today, packaging is a completely three-dimensional process, permitting the ideal integration of the development of initial styling geometries (main curves and guide curves), the generation of principal sections describing package and component envelopes through boundary surfaces into the overall process. Package functional surfaces are three-dimensional surfaces, describing the space occupied by individual components (e.g. diesel engine with its envelope surface determined for idle vibrations or the front wheel with the envelope surface of permissible steering and suspension movements, arm reach envelops of the passengers according to legal and ergonomic specifications or viewing pyramids.

50 months before SOP, the principal sections are created in 3D models using standard locations in the car coordinate system, and standard names for the geometric elements. To begin with, cross-sections of previous and other cars from the OEM product portfolio are used to obtain a coarse description of the topology of the zones and compartments to be developed. This coarse description is used to reach agreements and compromises between conflicting objectives with all departments involved in the car evolvement process. Subsequently, the principal sections positioned at standard locations in the car coordinate system are replaced by actual development sections derived from the functional surfaces of the package, from styling data and concept development results. These development sections are used for documenting relevant problems, reaching agreements between the design departments and for creating primary design surfaces.

In the third phase, approximately 40 to 30 months before SOP, the vehicle integration plan (see figure 5.4) is detailed further. In the middle of this phase, the package of the basic car model of the desired product family is officially released (package freeze). In the meantime, about 150 sub-assemblies of the car have been entered into the package plan. It is the main goal of the packing process to generate the geometric evidences that all legal and other requirements and specifications including OEM-specific regulations and standards are met and all conflicts resolved, and to keep the number of design variants to a minimum. Examples of such issues are the seating layout, luggage compartment layout, direct and indirect views of the driver, head clearance and head impact zones, accessibility of control elements, access to car, location and layout of restraint systems, pedestrian safety, bumper heights, ground clearance, slope angles etc.

With the aid of CAD and DMU (Digital Mock-Up), the packaging team continuously checks virtual design surfaces, module and part designs against hard points and envelope surfaces and possible collisions of components. This way, the configuration of the car is verified more and more in the 3D models.

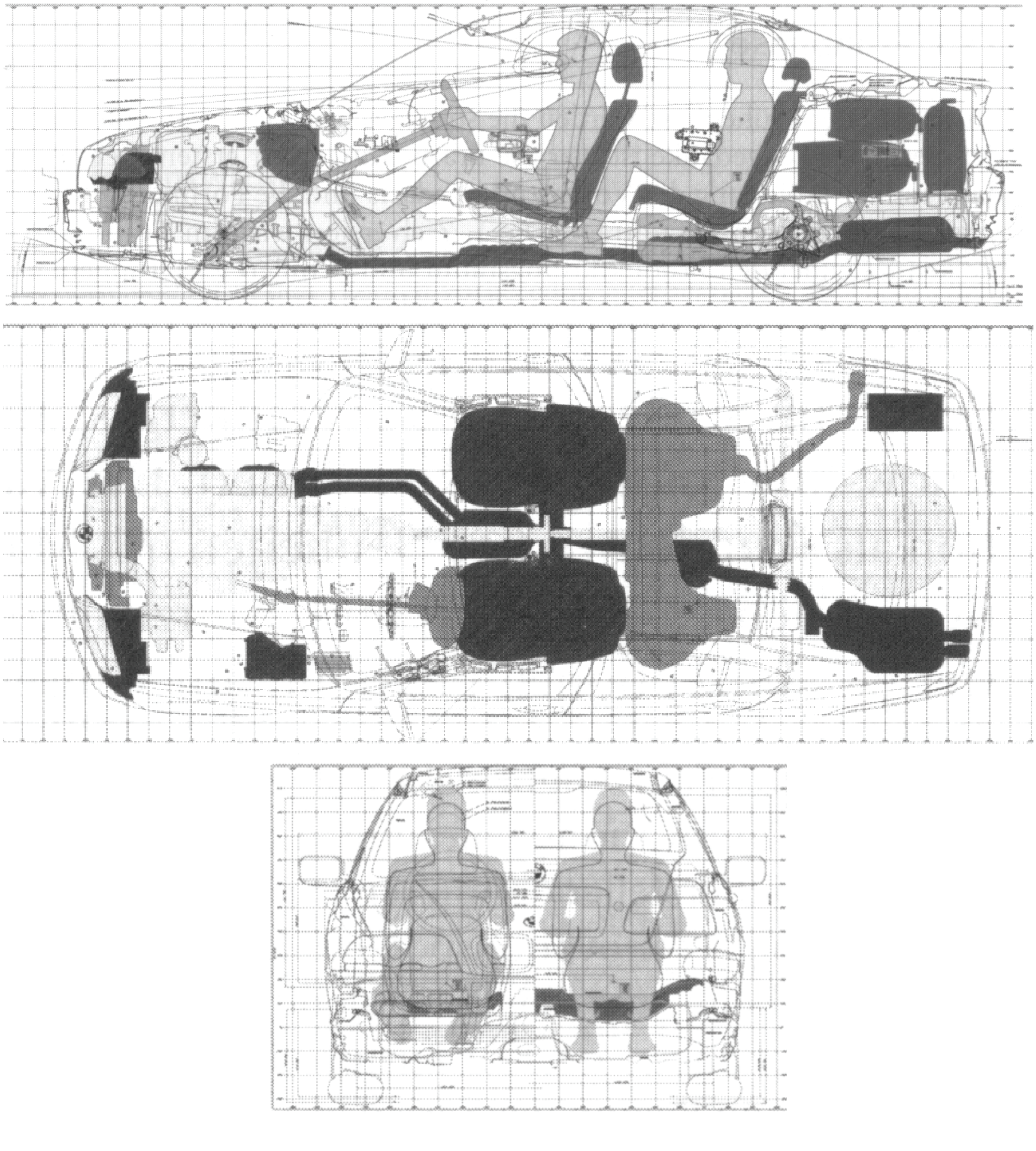


Fig. 5.4 Vehicle integration plan (side view, plan view, front view) (Grabner & Nothhaft, 2006)

In this phase, the first physical models of the future sub-assemblies are produced e.g. milled from hard foam. These physical mock-ups are used for experiments and test backing up the virtual validation processes, e.g. with ergonomic investigations performed by test persons, and with manufacturability studies.

After package freeze, the overall geometric data of the car is prepared according to internal standards and GCIE (Global Car Manufacturers Information Exchange Group), and simplified overall car plans according to GCIE are prepared for exchanging data with other OEMs (see figure 5.5).

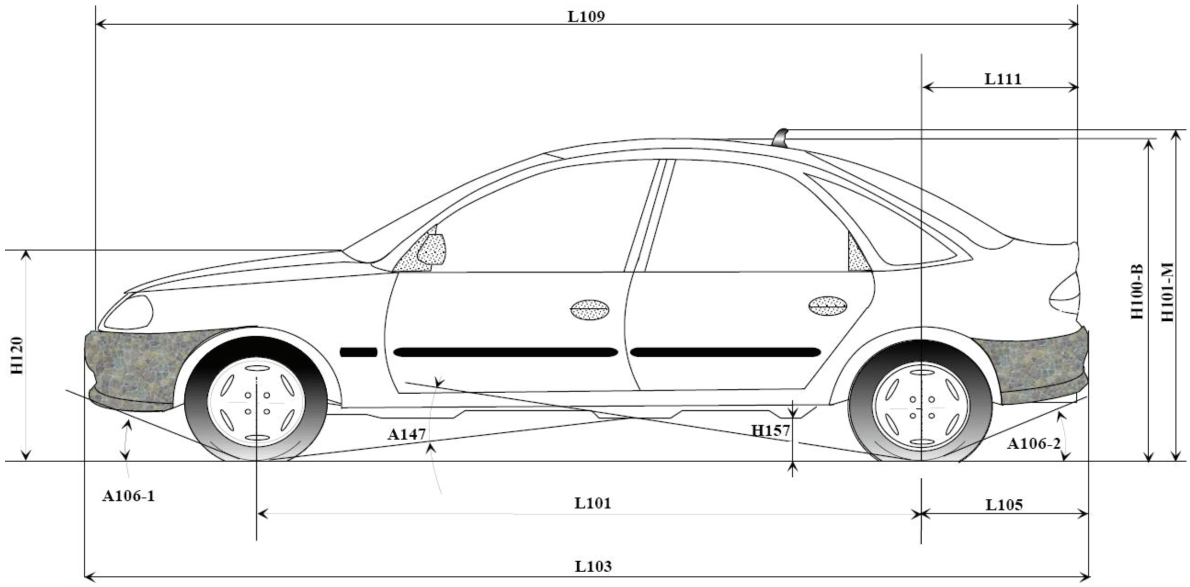


Fig. 5.5 Example of global standard package dimensions acc. to GCIE (2006)

Even during the detailing phase, design modifications and adjustments accounting for current research results may occur. Consequently, the packaging team remains responsible for the validation and verification of all such changes with respect to legal and other requirements. In mass production phase (detailing phase), the ergonomics team assumes the responsibility for the fine-tuning of control element surfaces and locations, operating forces and paths.

At the close of the serial phase, production part homologation processes and type approval are carried out, supported by the packaging team.

5.2 Technical Support of Aesthetic Design (Styling) Exterior and Interior

“The most important function in aesthetic design is to give the product a soul of its own. This soul is not alone created by technical development neither package or with the decoration; the product soul can only be created by a stylist who is able to combine aesthetic, style and emotion.

In the competition for market dominance the manufacturers (OEM) have to satisfy customer’s demands in ever increasing market segments. Parallel to the daily complex demands in customer lifestyle and the increasing number of products on the market, styling and “product soul” are growing in importance. You can’t just glue on “product character”, nor is it achieved by just using the logo symbols and colours of corporate identity. The combination of brand identity and “product soul” is achieved by the interplaying of aesthetic design, interior and exterior, of colours, shapes and materials. Thus aesthetic design defines the brand.“ (Ostle, 2003)

The aesthetic design process was the last manual layout process that has changed enormously through the developments in the virtual world. With the aid of computers it is possible to serve better process integrations and to have a greater variety of styling layouts in a shorter time span. Years ago the pure development of design ideas with manual sketches, renderings, tape renderings, clay (Plastilin) modelling, synthetic wood (Epowood, Uriol) or hard foam where on the forefront. Today the whole design process is developed in a new order with reduced numbers of format and media discontinuations aided by 3D computer aided styling systems with a great variety of import and design functions supported by large screen projections and 3D VR visualisation (Power Wall, Cave) (see figures 5.14 and 5.15).

Sketch boards including software for sketching, rendering and airbrush techniques are available for freehand sketching. It is thus possible to underlay a 3D package while using the 2D sketcher, which leads to the exact and correct proportions and perspectives. Or it is also possible to scan back ground photographs or handmade sketches to be used for a 2.5D styling process.

Today the way back from milled and hand finished design models into the computer (redesign) is possible by use of modern scan and import functions. High-resolution real time visualisations and the animation of the 3D design models enhance the efficiency of the styling process.

Aesthetic Designers are still sceptical toward the new process: “Digital models tend to have a more optimistic appearance on the screen and on power walls. Only the physical and tangible model will give you the best impression. It is true that the CAD models save a lot of money. It is however fact that a car factory sells real cars. Because of this fact it is necessary in the phase of deployment to work on real cars.” (Kraus, 2007)

5.2.1 Core Areas of Aesthetic Design (Styling)

“The exterior is love at the first sight. The interior is the marriage.” (Sielaff, 2004)

The basic form of organisation of aesthetic design is advanced design, exterior design, interior design and colour & trim. These areas are supported by modelling, studio engineering and studio management. Some OEMs assign ergonomics and the Class A surfacing also to aesthetic design.

Advanced design and concept design follow similar rules as early project concept developments of assembly groups within the technical design process. Both are often independent of a core project. The characteristic of concept design and advanced design is to collect ideas and visions without regard whether they can be integrated into the technical design process. Advanced aesthetic designers track down new design trends based on the information of trend research. The results of their creativity are translated to design models or even show cars for car exhibitions.

Exterior aesthetic design creates the specific visual character of a vehicle by designing proportions of painted panels, glass and light surfaces. Additionally character lines, highlights and graphic elements determine the visual character. Thus brand image is created by the typical styling of the front and rear end or the shaping of the C-pillar. Aerodynamics plays an important driving force for composition work of exterior for the last decades. Recently the legislation for pedestrian’s protection has influenced the arrangement of the front-end considerably during the last years.

Interior aesthetic design must also include haptics, scent and the auditory sense into the arrangement besides the visual language of shapes. Not only seating package, ergonomics and passive safety influence the interior process. Also topics like seat design and arrangement, accessibility of operating units, vision of instruments and many more are of big importance. The aim is to offer the customer a comfortable, functional interior and a safe and secure feeling. The selection of materials for the interior components influences this point largely (see figure 5.6).

In close co-operation with paint, textile and plastics industries, colour & trim develops colour and material combinations for interior and exterior, texture of plastic parts, fabric and leather for seats and trim etc., which have to correspond to future fashion trends. The careful selection of materials must also meet all requirements for quality and durability.

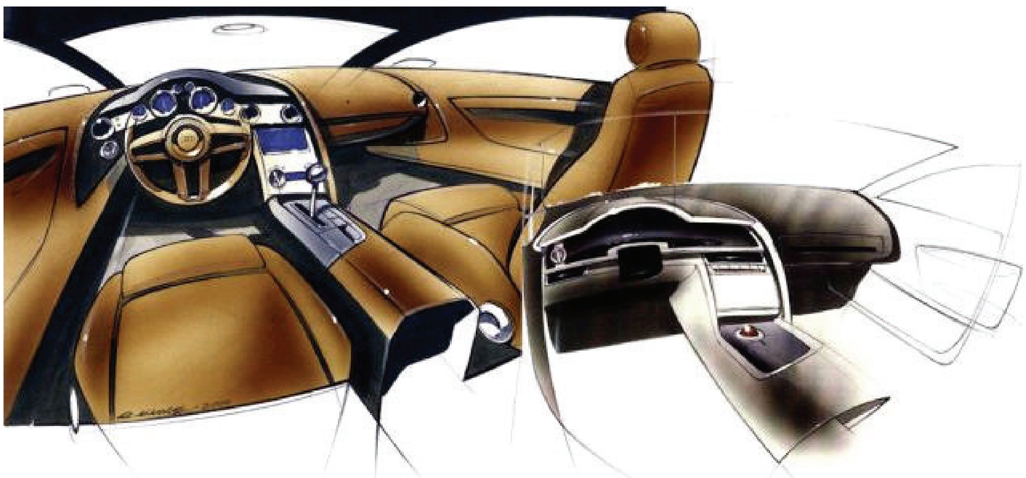


Fig. 5.6 Renderings of the interior (AUDI, 2008)

The roles of designers and modellers are often fusing during the aesthetic design process. In the conventional process modellers have largely been occupied mostly to convert sketches, renderings and tape plans into clay models. They were optimising it in close co-operation with the aesthetic designers. Because of there artistic skills and excellent capacity to think in three dimensions, modellers are today occupied with aiding 3D styling models on Computer Aided Styling system (CAS) work stations. Afterwards, the virtual styling models are milled in clay and are completed and optimised by the modellers. Then the virtual model is updated with the latest design in a redesign process by scanning and surface modelling.

Studio engineers play an important role within the design process. They mediate between the aesthetical and technical design process. Like the modellers studio engineers must be able to understand and interpret emotions and motivation of the stylists. Their task is to demonstrate the possibilities and the technical limits to the stylist. They are also authorised to represent the stylist’s claim to the engineers of the other development departments. Studio engineers are continual compiling the latest information about working methods, production processes, materials, legal requirements etc. and they are developing new technical ideas. The aesthetic designers get continuously informed by them about the possible innovations and consequences for the styling process. Studio engineers are responsible to guarantee the first functional feasibility of a styling by the use of principle sections, layouts and model building. They conduct the aerodynamic tests of design models. Together with method planners studio engineers consider about the production realisation at this early stage. As “ambassadors” of aesthetic design, studio engineers present the new styling to the other departments of development. They conduct the design and the development from the beginning of the PEP shortly until the SOP.

5.2.2 The new digital Styling Process

After a phase of discussion and interpretation of the list of goals of the new project the styling process begins with a contest of ideas. Every aesthetic designer involved displays his individual interpretation of the project in layouts and sketches. Within the styling process, which consists of creative phases, presentations, discussions and decisions, several competing aesthetic designs are worked out at the same time. At the end of this process there is one styling concept which combines essential details of these competing styles (see figure 5.7).

At the beginning of the aesthetic design process (see figure 5.7) the artistic and creative work comes to the fore. First of all it is important to let the design ideas ripen. The package / aesthetic design convergence process starts afterwards. Thus the artistic process of the tracked topic begins with freehand sketches of free chosen perspectives. Not till the second step these handmade sketches are digital edited with the support of sketch boards and are converted to the proportions of the underlayed 3D package. For that purpose the freehand sketches can be scanned to reshape and to change those on the sketch boards (see figure 5.6).

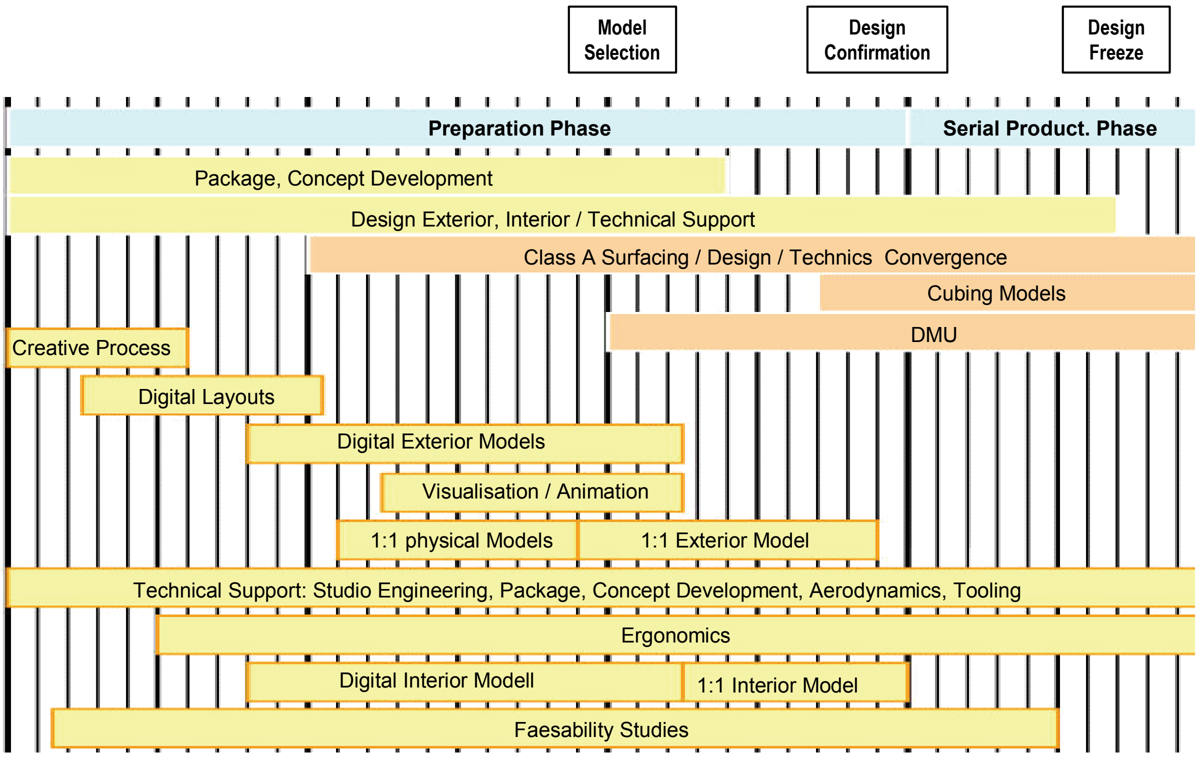


Fig. 5.7 Digital Aesthetic Design (Styling) process acc. to Ostle (2003), Dietze (2001), Damme (2003)

In the second phase of the aesthetic design process the conversion, the refinement of the proportions and the composition of details is carried out by 3D modelling in CAS. Orthogonal 2D outlines can be used as the basic framework for modelling and designing within the 3D model (sketch mapping). The stylists and modellers develop precisely tailored surfaces on the CAS system. The shapes and proportions of the model can be modified efficiently and the 3D data is available anytime so that a constant surface data flow to the development is possible. In the first instance five or more competing concepts are tracked. The concentration on two concepts is carried out shortly. Physical clay models of these two concepts are manufactured in a scale of 1:1. The precise refinement work takes often place at milled partial models, which can be added with assembly components from rapid prototyping or else of mass production.

Within the hand-crafted process renderings and tape plans have been the pre-stage for the physical model. Alongside the multifarious possibility of inspection and the possibility to animate styling models on a screen or a silver screen for visualisation and discussion, renderings (see figure 5.6) and tape plans are nowadays derived afterwards from 3D models of the CAS system.

The redesign process turns out to be demanding. The scanning data of physical models, which have been milled from CAS data and been changed and optimized, are interpretable and sometimes redesigned surfaces do not reflect the styling idea of the optimized physical models. After closing-off the second phase the aesthetic design of the vehicle has largely been detailed and one final concept has been chosen.

Fig. 5.8 Preparation of Styling adventure model at the end of styling phase (AUDI, 2008)

The finish of aesthetic design takes place within the third phase. The exterior model and the interior model merge to one model (see figure 5.8). Parallel to the second phase the Class A surfacing has already begun to optimise the surfaces of the design models in CAD. The definition of the gaps in position, width and producability as well as the quality of the surfaces has to be adjusted between aesthetic design and Class A surfacing together with the demands of the technical conversion. The designed

Class A surfaces are tuned with the aesthetic design by using physical data control models of the interior, exterior and the complete vehicle (cubing models) and are finally determined. Stylists accompany the technical design of the assembly components from the concept development till detailing phase. They ensure the accurate conversion of their aesthetic concept, especially in cases of the often required compromises within the development and the production preparation.

5.3 Class A Surfacing / Styling / Techniques Convergence

"Today the Class A surfaces are the geometrical representation of all surfaces that are visible for the customer both interior and exterior under consideration of all technological and shape aesthetical demands." (Lender, 2001)

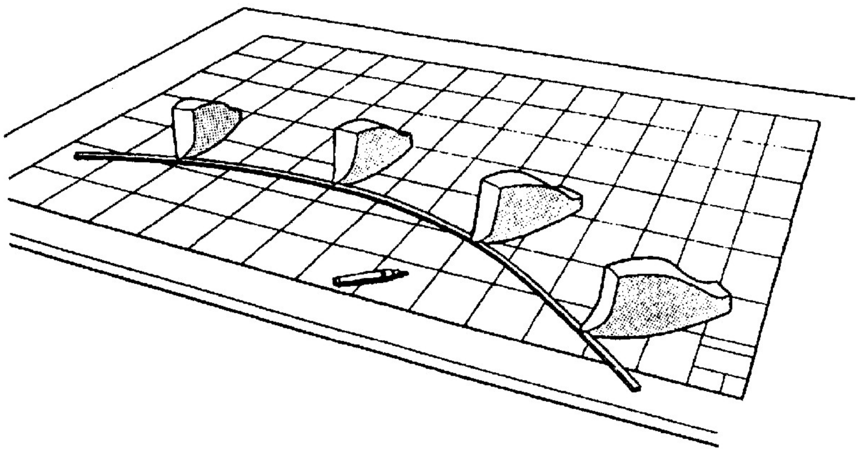


Fig. 5.9 Manual Class-A surfacing with spline and spline weights

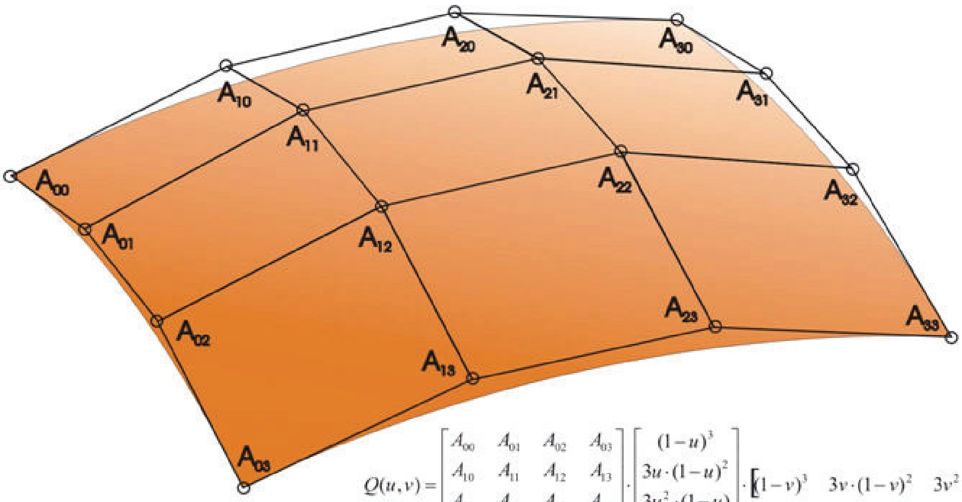
The former manual Class A process originally derived from shipbuilding. In the search for flow-enhancing designs of hull shapes it was possible to design grid sections (length- = vertical frames, breadth and height sections) through the hull under support of long, flexible splines, made of homogeneous material. These splines were kept in desired form by the use of spline weights (see figure 5.9). By alternately raising the weights curves with smooth curvatures developed which were free of dents, waves or any kind of inconsistencies. Thus surfaces were clearly defined along the sections however the areas between the sections were not defined.

Up into the nineties automotive Class A surfaces were designed by this wire frame method of grid sections and 3D curves. Today however without exception the Class A surfacing is completed with the aid of special CAD programs e.g. ICEMSurf or CATIA V5 workbench ICEM shape design. All of these programs are based on the principals of curve- and surface algorithms of de Casteljau and mathematics of Bèzier (see figure 5.10).

The Class A process is a major component of the PEP. It's primarily importance lies in the early position in the development process and thus the big influence of all the following processes which are continuously dependent upon up-to-date surface data. With the introduction of Class A surfacing into the early stage of PEP a dynamic loop process of surface and gap modelling and adjustment under support of all relevant specialist begins to harmonise the demands of styling and technical design. Class A development is completed with the release of the data control models which is followed on by the release of all customer relevant surface data of interior and exterior vehicle surfaces.

The process of surface development is based on scanning data which is generally available as a mesh of polygons which is originated from a cloud of scanned points. For better orientation the polygon mesh is added by relevant grid sections. The area that has to be developed is "divided" in mind into primary surfaces and secondary surfaces. Primary surfaces are those surfaces that are used as the principal or basic surfaces for the adaptation of further geometry e.g. offset surfaces or character curves. Secondary surfaces are those surfaces which are based on primary surfaces or which add further details to the primary surfaces, such as bent portions, blend surfaces or fillets.

E.g. contact points on each of the primary surfaces are used to hustle surface patches of low order (see figure 5.10) onto the primary surfaces. By careful and equal manipulation of the polygon points the surface patch will closely resemble the scanning data. The primary surfaces (patches = unbounded surfaces) are larger than the styling surfaces. Character curves can be achieved and smoothened from the scanning data or intersection curves can be gained in combination with other primary surface patches. The patches (unbounded surfaces) are bounded by the curves and become faces (bounded surfaces). Any discrepancies in the scanning data or in the styling model at hand will be interpreted by the Class A surfacing engineer to harmonise them with the aesthetic design. Surface data the Class A surfacing department provides includes all visible surfaces including first bent portions and first radii.



$$Q(u,v)=\begin{bmatrix} A_{00} & A_{01} & A_{02} & A_{03} \\ A_{10} & A_{11} & A_{12} & A_{13} \\ A_{20} & A_{21} & A_{22} & A_{23} \\ A_{30} & A_{31} & A_{32} & A_{33} \end{bmatrix} \cdot \begin{bmatrix} (1-u)^3 \\ 3u\cdot(1-u)^2 \\ 3u^2\cdot(1-u) \\ u^3 \end{bmatrix} \cdot \left[(1-v)^3 \quad 3v\cdot(1-v)^2 \quad 3v^2\cdot(1-v) \quad v^3\right]$$

Fig. 5.10 Bézier surface patch of low order (Schreiber, 2007)

Fig. 5.11 Comparison of surface and gap definition AUDI 100 (1986) and A6 (2006) acc. to Großjohann (2007)

The visible surfaces of the interior and the exterior are interrupted by gaps where ever component parts meet. Gaps are an import stylistic element for an aesthetic designer. While designing the gaps, functionality of movable components and highlights on surfaces have to be taken into account. The unbounded visible surfaces and the curves defined by the aesthetic design and safeguarded by technical design, are the basis for the Class A design of the gaps. For the development of a gap (see figure 5.11) the visual constant appearance of the gap, seen from the customer's typical point of view, is to be defined instead of defining a theoretically constant gap. In critical areas the surface position in the area of the gap is moved within height (displaced) and/or the theoretically arranged gap varies in breadth.

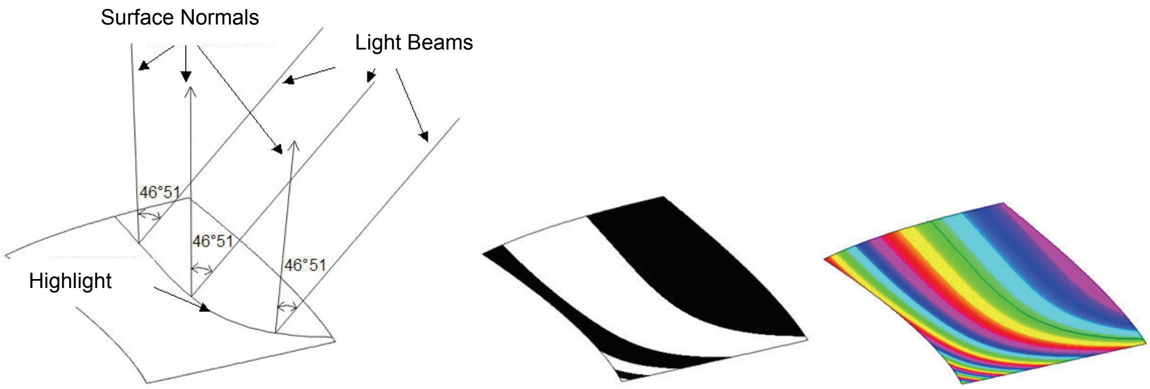


Fig. 5.12 Quality control of surface definition with highlights (Isopodan, Lines of constant brightness) (Petersen, 2003)

Fig. 5.13 Quality control of surface and gap definition with panel of parallel light sources LHS in VR (Gegalski, 2003) and RHS in laboratory (Dehn, 2001)

The virtual world offers a multitude of possibilities to evaluate the quality of Class A surfaces. Highlights are a common method used (see figure 5.12). In theory light sources are reflected under various angles to the free form surface. Lines of constant brightness (highlights, isopodan) are the geometric positions where the angles between light beams and surface normals are identical. A further virtual and even practical used method is the projection of a panel with parallel light sources onto the free form surfaces (see figure 5.13).

The Class A department starts with its work as soon as the aesthetic design reduces the number of the possible styling concepts to two. The styling models are milled and modelled in precise detail in Plastilin in scale of 1:1 for the first time. To be able to evaluate the models virtually and to make the surface data available to the following departments as quickly as possible, the two models are scanned. Within the Class A surfacing process concept surfaces are created out of the 3D cloud of points from the scanning. The concept surfaces do not yet comply with the final requirements because of their quality, curve progression and continuity. These first CAD models primarily act for the judgement and confirmation by the department of aesthetic design. Though the departments of technical design use these concept surfaces as the foundation for the assembly group design already.

The criterions to evaluate the Class A surfaces are:

- producibility,
- optical quality,
- compliance with package dimensions, concept coherence, carry over part (COP) concepts and design variants.

With the choice of the styling model (model selection) (see figure 5.7) by the management the number of exterior and interior models decreases to one. Shortly after that the design confirmation, including the incorporation of all open points and leadership issues into the model, takes place. The current development status is now harmonised regarding to package, concepts of technical design, aerodynamics, ergonomics and occupant safety. Hence the technical stage of maturation has proceeded to the point where the development of the final Class A surfaces begins. Therewith a process of virtual and physical data control models and master gauges begins which conducts the PEP till the SOP and which shall safeguard Class A surfacing and the following technical design of components (prototype parts as well as mass production parts). In an automotive project between 600 and 800 interior and exterior components, which are relevant for visible surfaces, get processed in Class A surfacing. Gray zone surfaces, which can't be seen until the customer opens a door or a lid, and a lot of surfaces of the interior are designed as functional surfaces within the concept development phase and are at last finished in Class A surfacing.

Fig. 5.14 Virtual validation of the interior at the power wall (BMW, 2003)

The design confirmation (see figure 5.7) means to "freeze" the current aesthetic design model. From now on it is no longer possible for the styling department to shape on the physical model and the design subject is completely assigned to the follow up CA processes. Class A surfacing is the leading department to make modifications and to visualise. At this stage all styling subjects are completely displayed. This includes for instance all mounting parts of the exterior, e.g. handles or trim strips as well as panels or electric components of the interior. Furthermore a detailed gap plan is defined which can be adjusted to a minor degree. Even so the stylists still have the possibility to place change requests within Class A surfacing. Henceforth the areas of aesthetic design and Class A surfacing closely work together with the aid of visualisation (see figure 5.14, 5.15) and CA tools.

The design freeze (see figure 5.7) is the next milestone within the Class A process. At that time the aesthetic design area has finished a styling adventure model (see figure 5.8) which contains interior and glass surfaces. All open issues, which were detected at the confirmation of aesthetic design, are incorporated in the Class A surfaces. This is the base for the digital data control model which assures the design and the visible surfaces within the virtual process. The digital data control model describes the final condition of the visible surfaces in every detail, for example varnished metal sheet areas, coloured and textured interior components or configuration variants. Thus it can also be used to safeguard the match up of colour

Fig. 5.15 VR validation of Class A surfaces (Haasis & Grebner 2007)

and material combinations (see figure 5.14). Weeks before the completion of the first physical data control model the virtual evaluation and the certification of all visible components takes place. Parallel to the Class A surfacing process a digital controlling of functions continuously takes place in all development areas. Functions, such as kinematics or over push effects of lids, the air distribution of the air vents or the head impact zones of the instrument panel and many more are checked in detail (see next sub chapter).

The result of the first physical Data Control Model (DCM = Feasibility Cubing) is a complete and full-scale milled model on the basis of CAD surface data. The DCM or else master pattern controls the Class A surfaces, it resembles the design model and represents the decision made by executive committee. Today the DCM is created by high-performance milling machines which allow varnish capable surfaces. The assembling takes place on basis of synthetic wood block material (Uriol) supported by aluminium beams. Final surface refinements demand accompanying Class A surface design operations and lead to the point of release for the last milling loop of the final DCM.

Fig. 5.16 Functional Cubing model interior (Beutenmüller, 2008)

The functional cubing models of exterior and interior (see figure 5.16) complete the safeguarding of all Class A surface data. The body in white (BIW), all mounting and trim components of interior and exterior are displayed as a build-up which is geometrically equivalent with the final product. All exterior and BIW components are made from aluminium. The interior is displayed by Uriol or laminated components. Like on a real car doors and lids can be opened by using realistic kinematics. Also the original boot is displayed. Cubing models of mounting parts of exterior and interior are fixed the same way than in mass production. For example the functional cubing interior is verified to check if the mounting and assembly components are able to be installed at their fixing points, at the clips holes and if these fit together. The executive committee affirms the complete field of visible surfaces of the automobile including secondary surfaces as well as visible grey zone surfaces between interior and exterior.

5.4 Digital Prototyping and Digital Mock Up (DMU)

The development of a product roughly takes place within the phases of design and simulation, laboratory work on physical models and road testing (Figure 5.17). Calculation and simulation leads to an early safeguarding of design data. It also allows physical prototypes with a high maturity and an optimised testing. Alongside the early safeguarding by simulation, the testing of physical prototypes within the development process is indispensable (Breitling, 2007).

Digital Mock Up or DMU is a technology that allows product design engineers to replace physical prototypes with virtual ones, using 3D computer graphics techniques and configure complex products and validate their designs (Wikipedia 2007:3).

The term DMU, the digital build of prototypes, does no longer describe the full bredth of tasks for the support of visualisation and simulation within the PEP. Within automotive industry terms like Digital ProtoTyping (DPT) or else Digital Engineering Visualisation (DEV) are in use.

In automotive body design several verifications of the 3D Geometry take place. In former times verifications were performed under support of extensive 1:1 mock ups (e.g. seating mock up for ergonomic examinations or mock ups built from deep drawn plastic sheets to verify the assembly of BIW parts). Today we find a mixture of CAD driven, PDM driven and special isolated applications for DMU. These DMU

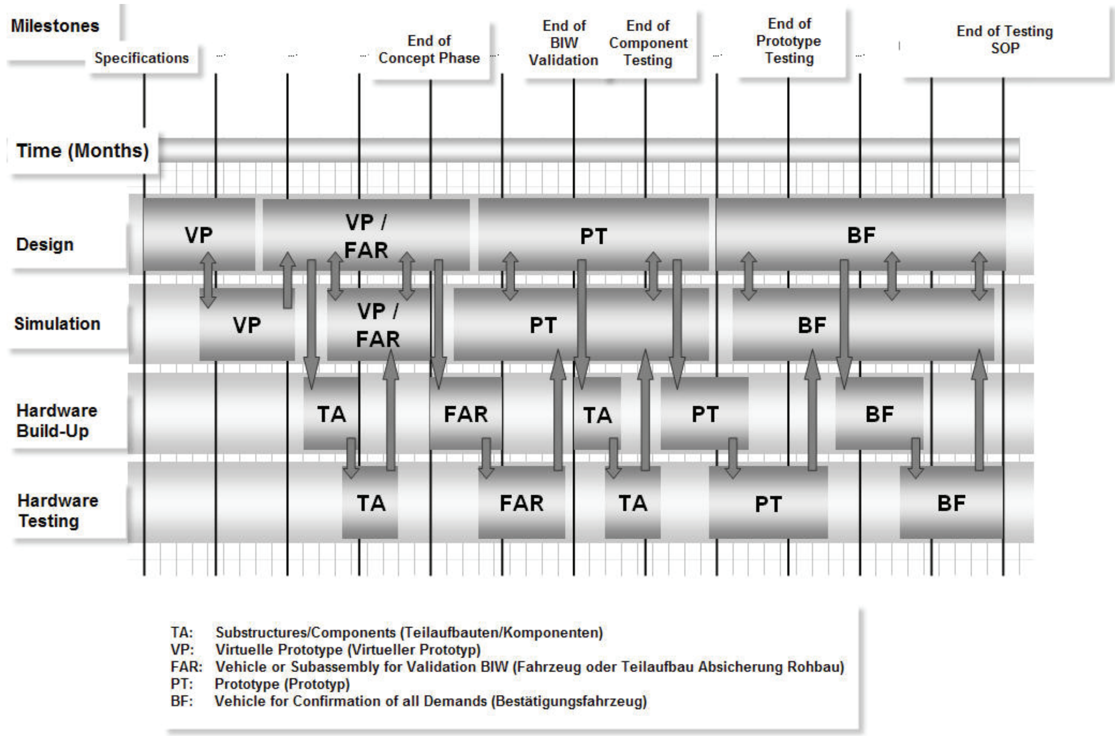


Fig. 5.17 Mixed Reality for the Validation of Passive Safety (Kötteritz, Daimler)

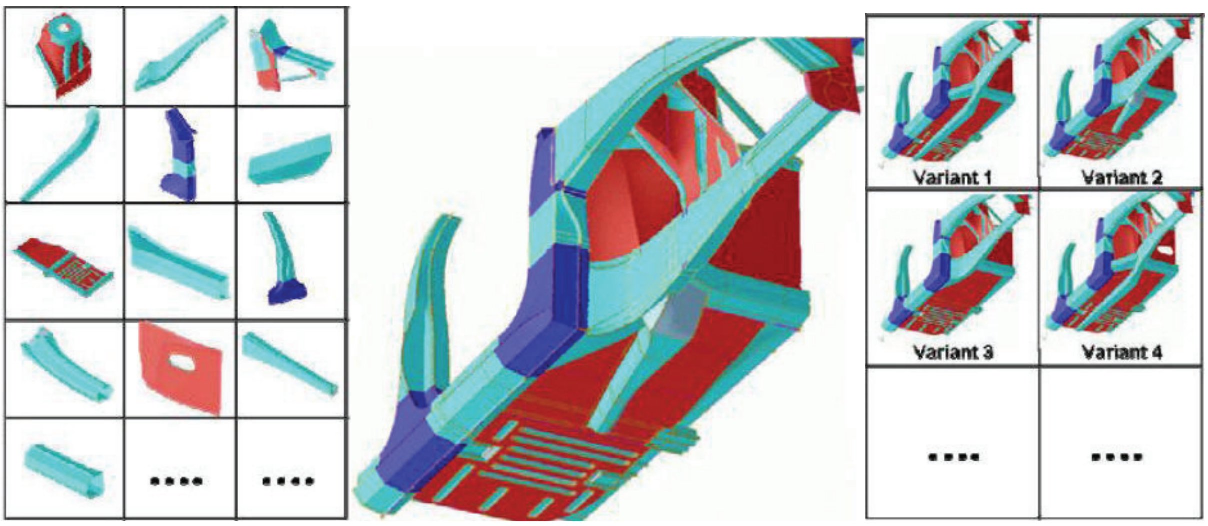


Fig. 5.18 Building set for early structural evaluations with sub and main assemblies (Hänschke, 2007)

supported technologies evaluate diverse development concepts of the early phase and often can early validate different design variants. Today the following fields among the application areas of visualisation and simulation are important for the safeguarding of body design:

- package lay out and homologation (see figure 5.19)

- visualisation of design data and Class A surfacing (see figure 5.15),
- aerodynamics (see figure 5.20),
- structural load cases,
- kinematic functions of mounting parts,
- deformation of parts within production or in common handling,
- tolerance management with its effects on surface and gap design as well as within the production (see figure 5.22),
- assembly, disassembly and handling / filling of assemblies,
- all crash load cases,
- durability of load-bearing components,
- noises, vibrations and harshness (NVH) a passenger feels,
- thermal functions of the heating/ventilation and air conditioning unit (HVAC),
- thermal comfort inside the car (see figure 5.21)
- loads caused by cataphoretic painting process
- and many more.

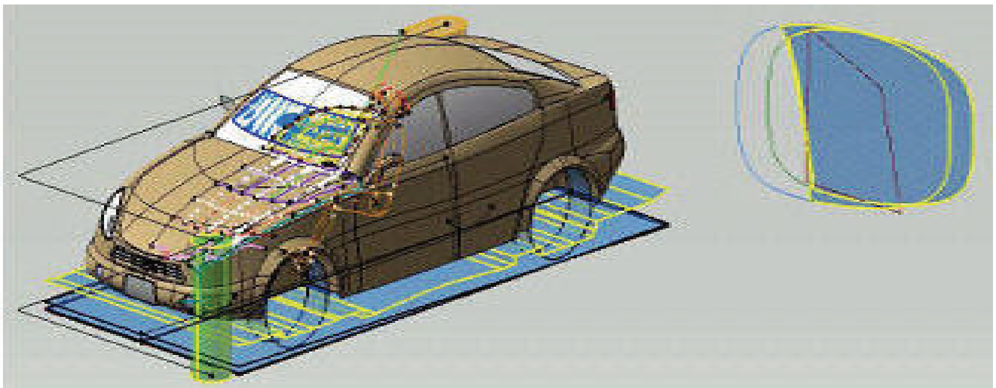


Fig. 5.19 CAVA-Software for package and homologation (Potthoff, 2007)

Forsen (2003) classifies the activities of DMU into four main areas:

- "GI: Geometrical Integration: All CAD-driven and powerless verifications belong to the area of steric geometric applications, such as: Interference Checking, Available Space Analysis, Cinematic Clearance Checks, Tolerance Simulations, and Ergonomic Examinations etc."(Forsen 2003) The Geometrical Integration already includes several functional integrations such as ergonomics examinations, kinematics of closures and other adjustable body assemblies, lighting etc.

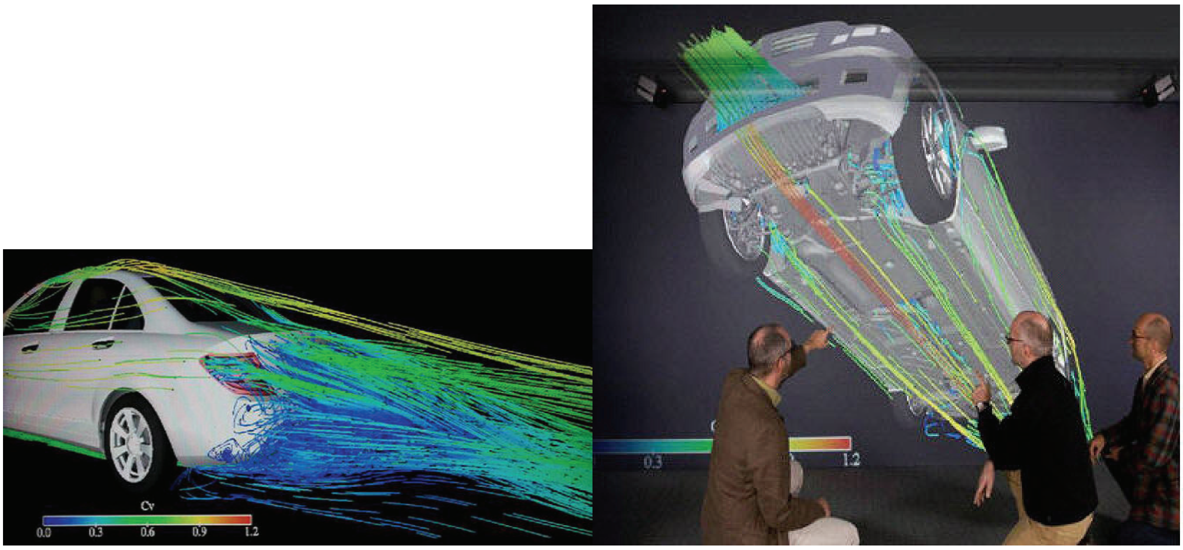


Fig. 5.20 CFD analysis (Breitling, 2007)

- "FI: Functional Integration: The area of functional integrations includes all power integrated calculations in the field of DMU, such as: Crash Simulations, Stiffness Calculations, Aerodynamic Simulations, and Driving Dynamics Calculations etc.

- PTI: Production Technology Integration: The term of production technology integration includes all verifications to safeguard manufacturing processes, such as Installation and Dismantling Studies, Deep Draw Simulations, Simulations of Jigs and Fixtures or Spot Welding etc.

- EI: Electronic Integration: A strongly growing rate of electronic components in vehicles intensifies the poor design of mechanical components up to the handling of complex functional contexts, controlled by electronic devices. DMU therefore must be enhanced by simulation methods for electronic integrations." (Forsen 2003)

Very important subjects within development and analysis are the design stages (status management) and design variants (variants management) which can only be organised for the digital calculation and simulation process under support of intelligent CAD software (PAD) and/or PDM systems. It is typical to analyse and rate a large number of alternatives within the early project phase. The main part of DMU data is taken over from 3D CAD data. The DMU simulations listed under Geometrical Integration (GI) can be verified in the CAD programmes themselves. Visualisation and rendering of large assemblies or dynamic cross sectioning often is handled within PDM systems. PDM integrated applications also support verifications in a development process with multi-CAD use. The special solutions of the functional integration (FI) often have special data formats which may course problems with the feed back of data into the design processes (Hagenah & Klar 2006).

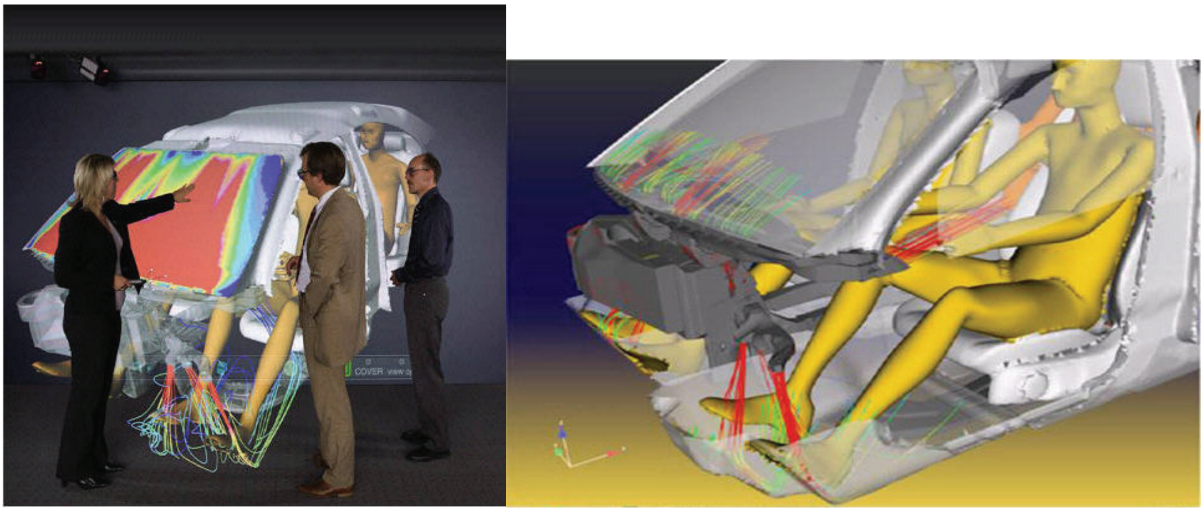


Fig. 5.21 Analysis of air and temperature distribution under different weather conditions using virtual imagery (Breitling, 2007)

All times the amount of data confines the actionability of design and simulation. The DMU data format JT (Siemens – Unigraphics PLM Solutions) has asserted the behalf of the digital development process (Pütz, 2007). Besides tessellated data with several levels of detailing, e.g. to visualise design data effectively, the JT format delivers exact geometric data (NURBS), e.g. for measurement aspects, after the conversion. It also delivers metadata such as the product structure or attributes like dimensions, tolerances, (Reference Point System) RPS points or welding spots. Thus it is possible e.g. to represent the complete environmental geometry within JT format while designing an assembly unit in a complex environment. The transmission of design history between OEM and system suppliers or different design departments is not needed.

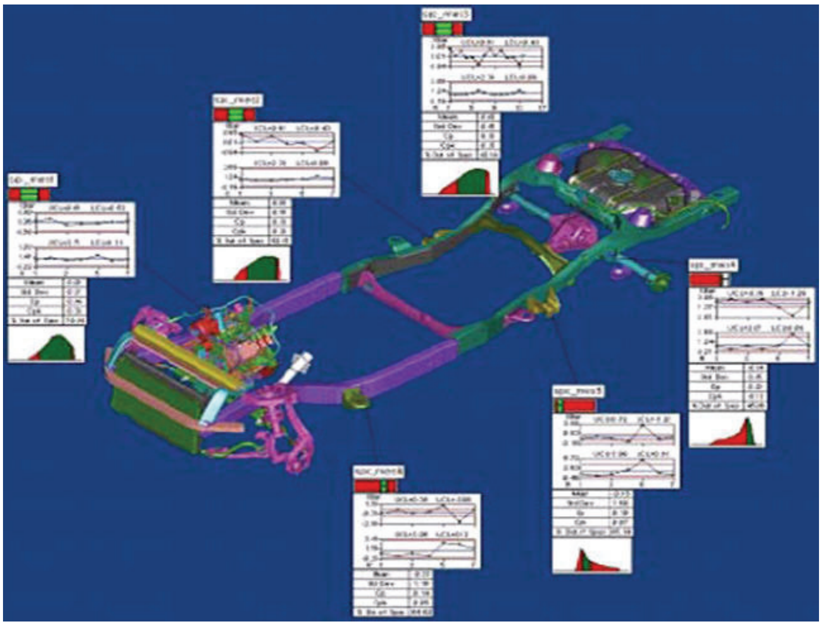


Fig. 5.22 Statistical Tolerance Analysis (Pütz, 2007)

Analysis of the complete vehicle with configuration and creation of DMU data in real time is not possible nowadays because of the high volume of data and different data formats. For these verifications DMU data are collected external from data collected and converted over night into JT or other exchange formats. The preparation of FEM meshes takes based on this exchange formats cannot be done completely automatically and takes another one to three days for an assembly up to several weeks for a complete vehicle. The results of these verifications can therefore not be up to date and often not deliver all versions (Hagenah & Klar 2006). The evaluation of constructed space in “Design in Context“-applications requires a connection between DMU and BOM-systems (PDM). Modern parametric associative CAD systems with connection to PDM system took over main duties from DMU: to safeguard the continuous consistent management of versions and the mainly interference free development of assemblies.

5.5 Concept Development / Design of Closures

The design of geometry and safeguarding of functions in concept and detail development is performed in several phases and interrelated in diverse ways (see figure 5.23). According to the product vision (catalogue of goals) declared, design and simulation work is divided from the whole vehicle into the single part of the vehicle body and reassembled several times.

For the concept and package phase as a general rule specialist from the different areas of the vehicle body development, from package, styling, design and simulation are concentrated in a project team to develop, organise and decide about development work for the basic approach within a short distance and to inform their centres of competence directly. Innovative concepts and new technologies developed independent from a special project are adapted to the new project in this phase. For every goal several alternative concepts are explored and developed to one optimum. While the suppliers and engineering suppliers were involved in the detail development in former years only, they are often integrated from the early phase of concept development now. Project management of design and simulation volumes of work is often performed by experienced body designers of the OEM.

Detail development takes place under supervision of the centres of competence responsible for the special areas of vehicle body. Most part of the engineering work is done by engineering suppliers and suppliers. During this phase parts and assemblies are developed in detail and their detail functions (of e.g. weight, producebility, comfort, crash etc.) are further optimized. While in concept phase simulation for the safeguarding of functions is executed on the basis of unripe CAD models, analogues models or CAE modelled geometry in the detail phase simulation safeguards the development on basis of ripe CAD data. But even in this phase new cognitions (laws, competition situations, mistakes in early phase etc.) can lead to a switchback into concept development. As the project steps are appointed definite until SOP expenditures are multiplicated in such situations.

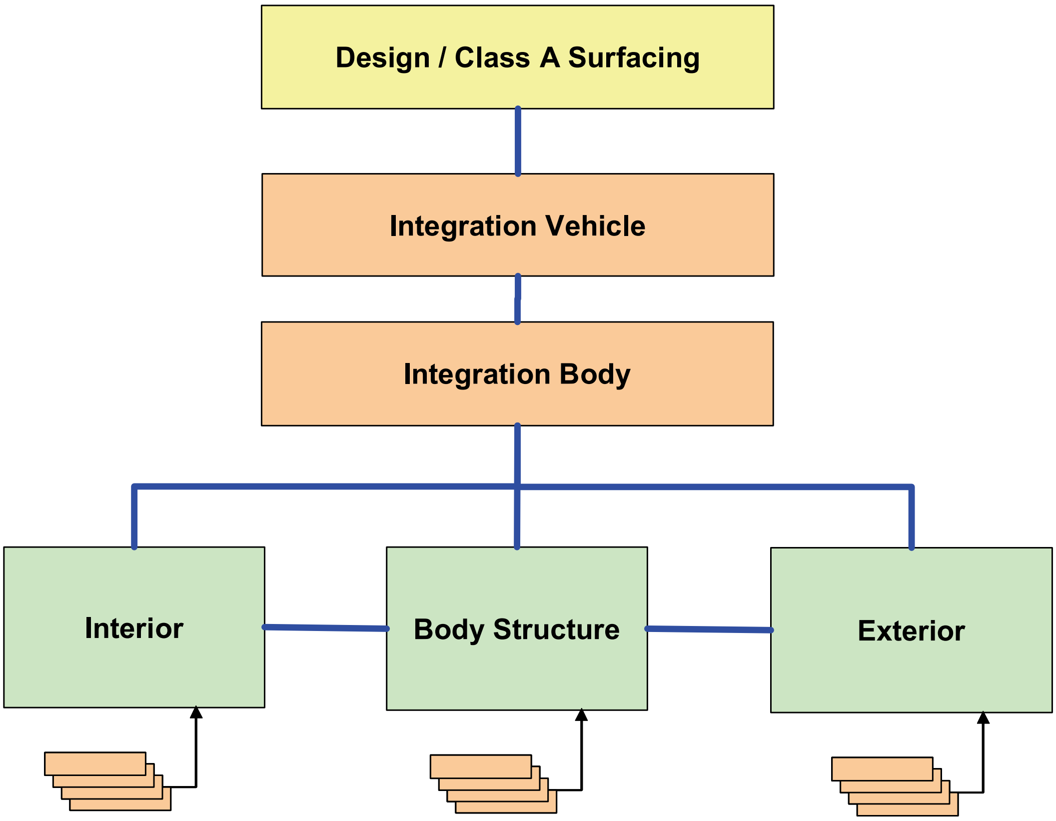


Fig. 5.23 Interrelated vehicle development

The activities described in chapter 5.1 – Package, 5.3 – Class A surfacing and 5.4 digital prototyping will be interpreted in this chapter on the example of side doors. Even here the explanations can deliver only a small view into the multifarious field of closure development.

Side doors as well as front and rear lids belong within the system "vehicle body" to the sub system "mounting components exterior" (see figure 5.24) while the structural parts of the vehicle body define the sub system "body structure". In the organisational structures of some OEM the interior trim of closures is also part of closure development.

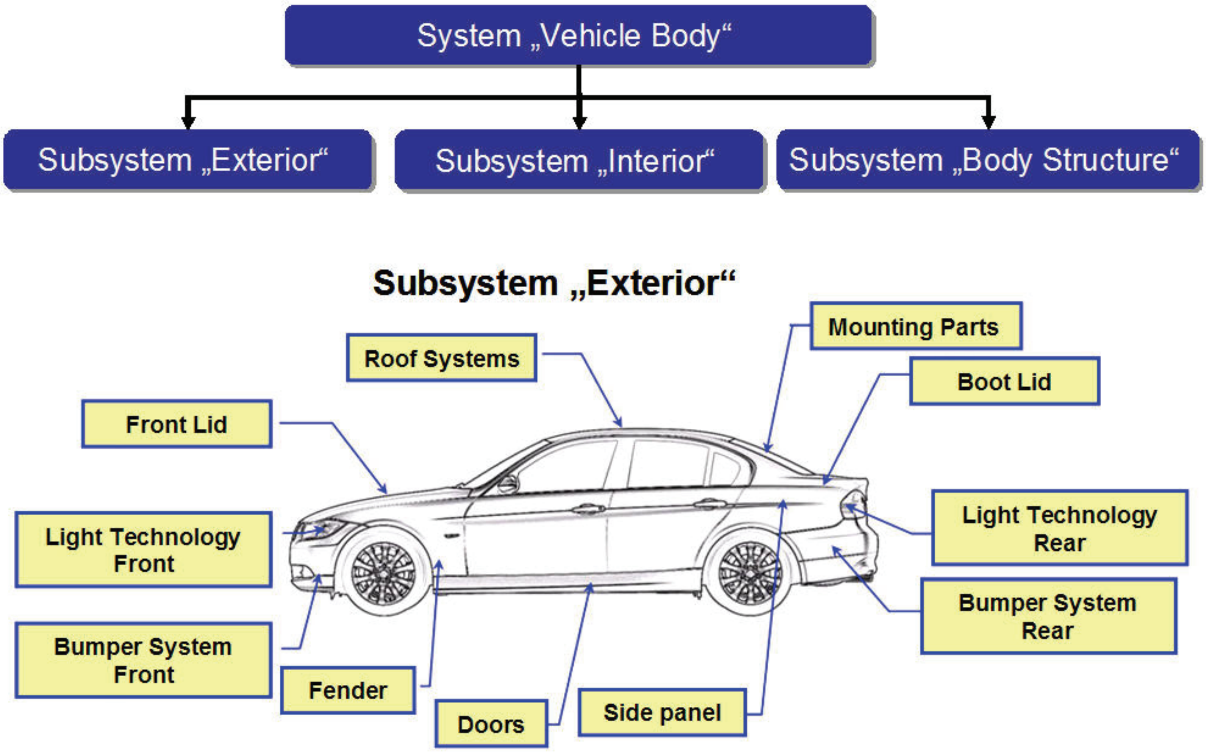


Fig. 5.24 Sub system “Exterior” in major system “Vehicle Body” (Haslauer, 2006)

- A closure is a device that allows access to a compartment.
- E.g. the access to the passenger compartment, the boot, engine compartment etc.
- A door protects the passenger compartment against climatic influences, noise, unauthorized access etc.
- Important elements of e.g. a swinging door are, besides the door structure, the hinges, the latch and the sealing gaskets.

The side door is one of the most complex assemblies of the vehicle body and must fulfil lots of requirements. In this chapter requirements are described which can be categorized as follows:

- Handling functions,
- Passive Safety,
- Quality impressions,
- Misuse and
- Durability

Side doors are designed according to different concepts. During the last years the frame door concept was established in mass production (see figure 5.25).

Handling functions are functions which have to be guaranteed for the daily use of the door. Nowadays handling comfort and safety play an increasing role.

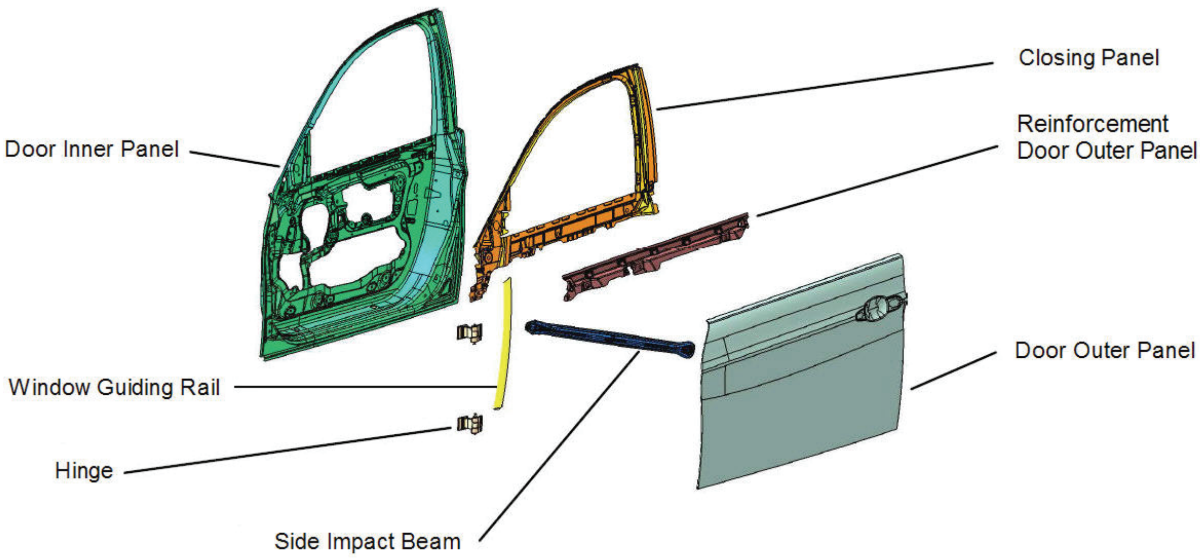


Fig. 5.25 Frame door concept in steel layer built design (Leinweber, 2007)

Legal requirements, directives and specifications (e.g. ECE, FMVSS), the crash test procedures defined by consumer organisations (e.g. Euro NCAP, IIHS), (CARHS, 2007) as well as in-house rules and regulations for Passive Safety are multifaceted and influence the design of side doors and side wall structures extensively. Side doors must support the safety of passengers at both side and front impact. And in all cases side doors must guarantee the rescue of the passengers after an accident. As examples from a multitude of designs the waist rail of the door which defines a load path for frontal impact (see figure 5.26), the side impact beam, the side airbags in the doors or padding arrangements in the door trim can be appointed. A lack of quality comes up when e.g. during polishing, the elastic deformations of the outer door panel lead to a noisy oil canning tendency. This effect and plastic deformations are not allowed. Other examples for quality issues are the stiffness of the arm rest and the map case or the door closing noises.

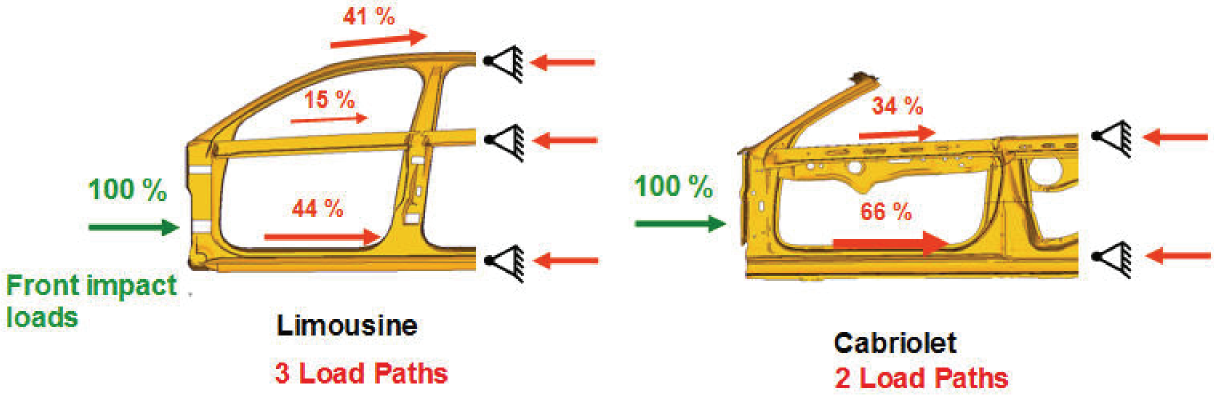


Fig. 5.26 Load paths in door and side wall structure due to front impact (Schulte-Frankenfeld et al, 2002)

Misuse loads are defined as soon as normal handling loads exceed the expectations. Doors must bear misuse loads up to certain limits. Examples of such conditions are

- the load case door sag. Here, a load is defined that represents a heavy person leaning on the door body.
- When a third person tries to get into the car by forced pull on the partially opened door window (see figure 5.27).
- When door is slammed with partially opened door glass with up to 30g.
- When the open door is over pulled, this extremely stresses hinges, door stop, door panels and pillar.

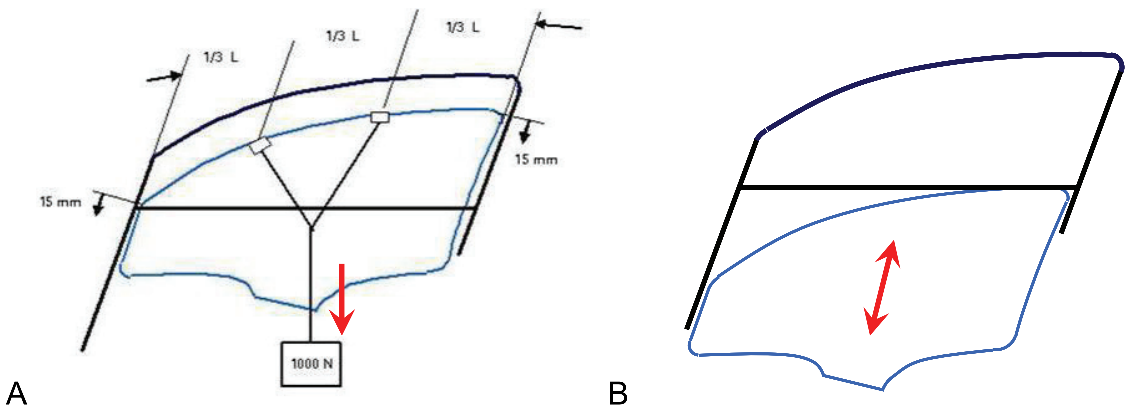


Fig. 5.27 Load cases at drop window: A: Forced entry; B: Opening and closing with / without stopper (Badri, 2008)

If the occurring loads do not exceed regular working loads, but occur in a great amount of cycles during the vehicle life span durability load investigations are necessary. Typical examples are the endurance tests “door opening and closing” or “window opening and closing” where wearing parts like sealing gaskets, latch, hinges, door stop etc. are examined.

The following examples show some design requirements in more depth:

5.5.1 Hinge Axis Positioning

The hinge axes of passenger cars are positioned within the side wall. This leads to the situation that a part of the door turns into the side wall while the door is opened. The depths of turn in amount depend on the distances between outer surface and hinge axis as well as front corner of door and hinge axis (see figure 5.28).

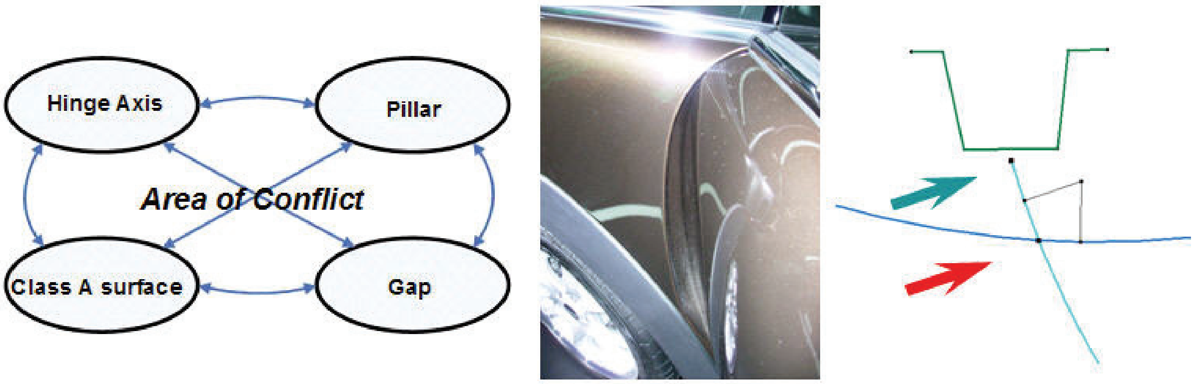


Fig. 5.28 Conflicts during gap and hinge axis definition (Picture LHS: Fischer, 2006)

The hinge axis is inclined marginally to lift the lower rear corner during opening and to allow comfortable opening and closing moments. The position of the hinge axis is defined by the curvature and inclination of the outer surface, the size and distance of the hinges, the general design of the door panels and connections and the adjusting ranges for tolerance compensation. The hinges used are mainly Carry Over Parts (COP). The mounting of the hinges parallel to the planes of the vehicle axis system allows the adjustment within all three directions of the coordinate system. This is necessary to adjust flush outer panels and parallel gaps.

Doors are nowadays often reassembled after painting, to enabling separate final assembly of the door. Hence in most cases the hinges are designed according to a two shear connection to unhinge and hinge the door. The travel needed for the unhinge operation has to be taken into account when the door gap is defined.

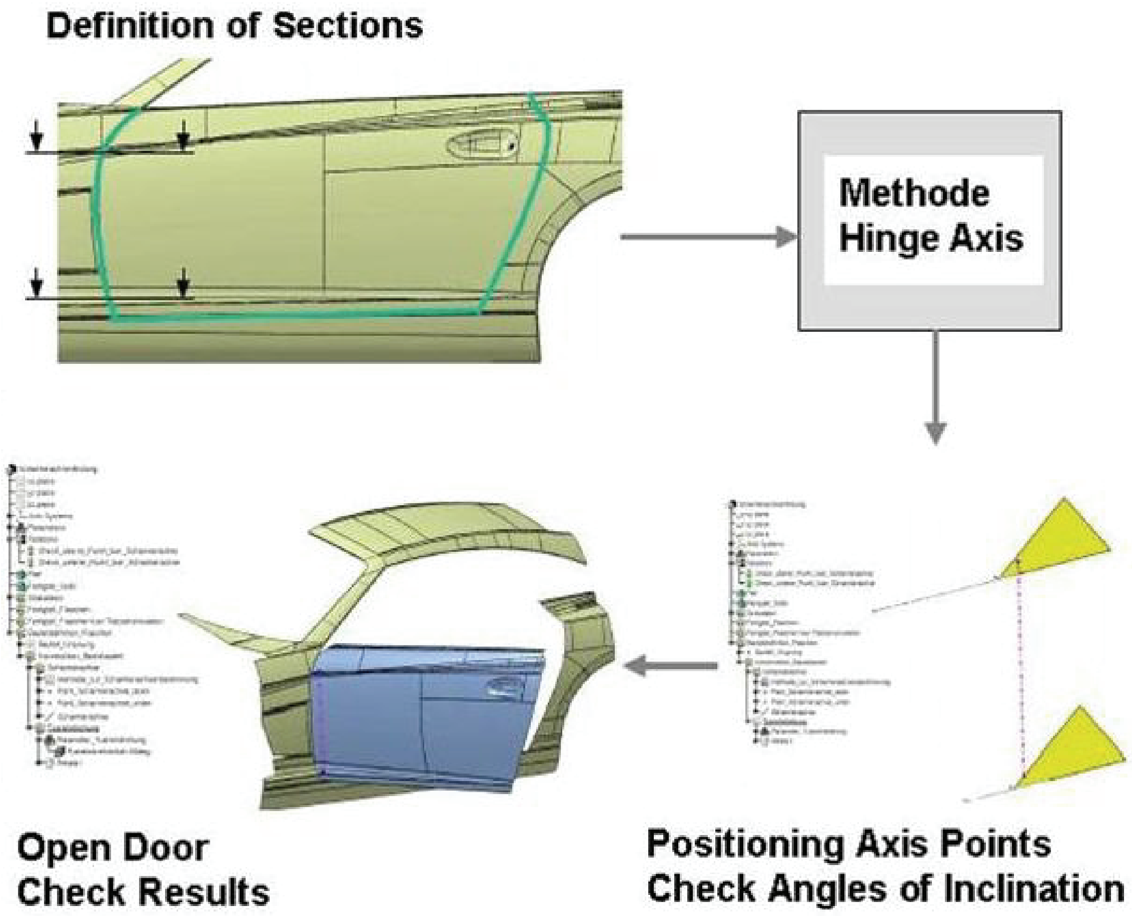


Fig. 5.29 Triangular corridor for positioning of hinge axis (Brockmeyer, 2005)

The door gap in the hinge axis area has to be safeguarded several times during the development process. As soon as the first 3D styling surfaces have been defined the studio engineers can check in critical positions on basis of principle sections if the gap defined by the stylist allows opening the door. Or the studio engineer defines limiting points and curves for the stylist to show a bandwidth for the definition of the gap. The first limiting curve and possible front corner of the door is defined when the outer surface of the opened door intersects the outer surface of the closed door at a defined opening angle (see figure 5.28). The second limiting curve for shaping and positioning of the outer pillar surface is defined by the front corner of the opened door. The more the stylist stays away from the first limiting curve the more the front corner of the door turns into the side wall and prevents a suitable side wall structure.

In the concept development the hinge axis is safeguarded again on basis of Class A surfaces and gaps. Under consideration of the parts mentioned above and critical tolerance positions a three dimensional corridor for the final position of the hinge axis is defined by three limiting surfaces on basis of the styling gap (see figure 5.29).
From the first concept development stage of the door to the completely detailed door the designs of the parts will be optimized in several loops. This will lead to several adjustments for the safeguarding of hinge axis and gap.

5.5.2 Latch Positioning

For a low noise emission and wear out of the latch positioning the latch plane has to be defined normal to the turning circle of the door (see figure 5.30). The second angle of inclination of the latch plane is defined parallel to breadth (XZ) sections of the basic slanted surface connecting the inner and outer side wall in the latch area. While in former designs the latch often was positioned perpendicular to the plane view this is the best way to allow the smallest possible constriction of the pillar.

The basic slanted surface of the door connecting the inner and outer surface of the side wall is defined by the travel line of the door glass and the slanting angle necessary to open and close the door. While this functional surface is defined planar at the beginning of concept design all surfaces of the grey zones except mounting surfaces have to be designed slightly convex at a later stage for manufacturing and aesthetical reasons.

The height of the latch depends on the question whether the door outer handle directly or indirectly fits the latch. The latch and all its components are positioned within the lower door case. For the operation between latch and striker a slot is necessary in the corner between inner door surface and basic slanted surface of the door inner panel and in some designs in the inner door trim as well.

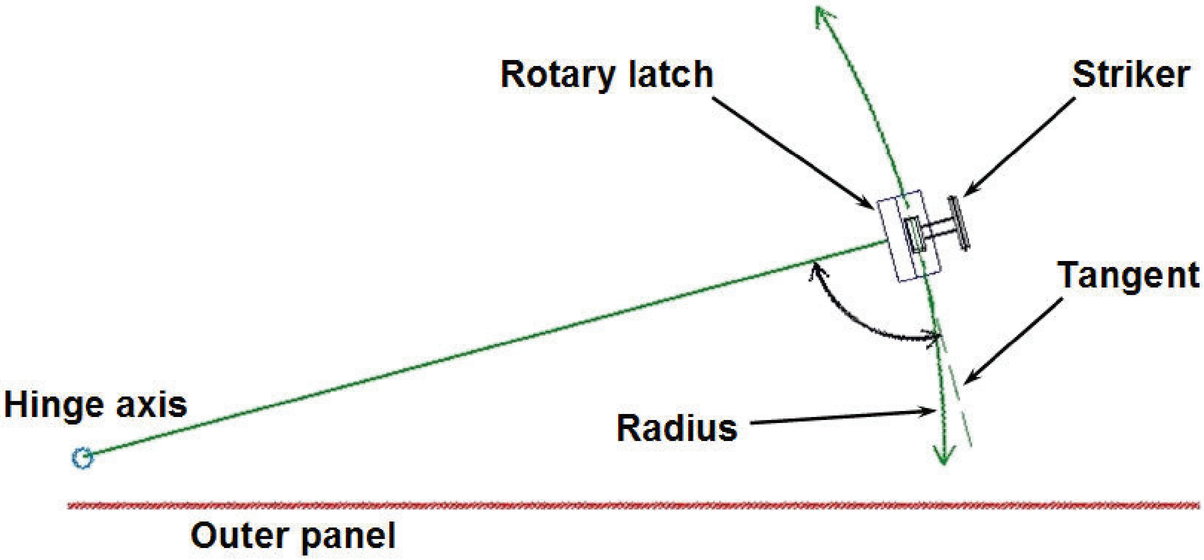


Fig. 5.30 Positioning of latch tangential to turning circle of closure

Fig. 5.31 Interaction between pull down (travel) line of drop window and recess of inner panel in latch area

To allow the movement of the door glass there must be a recess in the basic slanted surface (see figure 5.31). The size of modern latches with all their comfort functions need large recesses on the doors and corresponding pillar surfaces constricting the pillar structure. Some OEMs define the recess and its slope surfaces defined in figure 5.31 local only from the waist line down to the sill.

Optimisations of the side wall structure according to Passive Safety requirements often lead to changes and redesigns in the latch area.

To guarantee the hinge and latch functions dependent from the locking position of the latch legal requirements define longitudinal and transverse forces. These forces are simulated in designed position in a ridged portal and door environment as soon as first concept geometry for hinge and latch position is defined.

5.5.3 Drop Glass

When the side door area is defined in styling it must be approved whether curvature and inclination of the green house area correspond with the dimensions of the lower door case. On basis of longitudinal sections (cross sections, ZY) and principle sections of the package the size, shape and positioning of the door glass is safeguarded (see figure 5.32).

Has the door glass of former cars been plane or curved cylindrical most door glasses are part of a large rotational surface nowadays. Two or more longitudinal sections through the measured data of the styling model are the basis to define the rotation axis. As the rotational surface made from more or less elastic glass must pass the slot between the waist rails, limit ranges and combinations for the curvatures of the glass normal (R 1000 to R2500) and radial (R 20000 to R 150000) to the rotation axis are defined from experience (Pusilo et al, 2001). Modern CAD programmes allow the mathematical optimization of the door glass according to points of interest on the styling model.

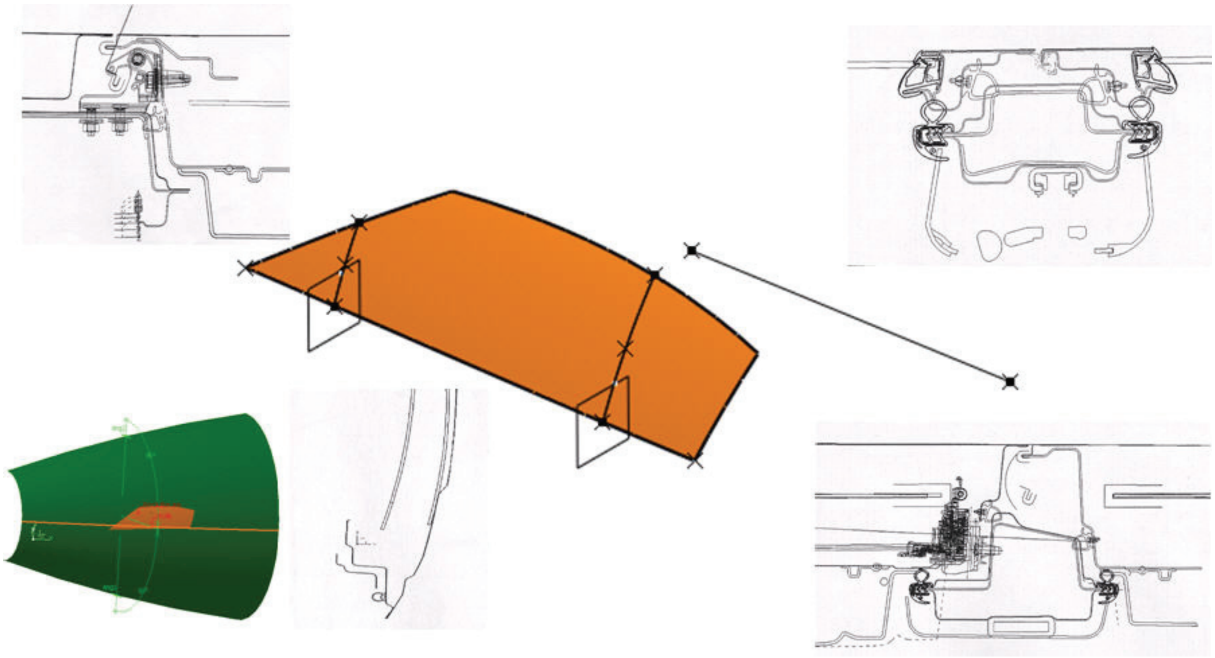


Fig. 5.32 Definition of rotational surface and four-side-trim of drop window acc. to Sonnenberg (2006) with principle sections acc. to Pusilo et al (2001)

The door glass of front and rear door is guided along the B pillar. The inclination of the front and rear corner of the B pillar defines a movement of the door glass combined from rotation and translation. The daylight area of the door is enlarged in the waist area by supporting and adjustment surfaces. Form and positions of the door glass must match up with the panels of the door e.g. in the sill area and the mounting parts such as door stop movement, side impact beam, latch etc. Minimum distances between rigid, flexible or moving parts have to be considered.

When the first concept design of the door panels is finished the door can be simulated and optimized with the following load cases:

- Door glass opening and closing with and without stopper in opened position,
- door slam with door glass partially opened with an acceleration of more than 30g (Daimler 60g) or
- forced entry through the window with a pulling force of 1000N on the slightly opened door glass (see figure 5.27).

5.5.4 Door Frame

The door frame defines the upper corner of the door. The nearly flush door glass is guided and sealed on the door frame. The sealing gasket situated on the welding flange of the side wall structure seals against the seal surface of the door frame. The sealing gasket situated on the door frame seals against the seal surface on the side wall structure.

The door frame of the frame door concept (see figure 5.25) is build by a deep drawn door inner panel designed from sill to roof rail and includes the cut out for the door glass. Combined with a deep drawn closing panel reaching from roof rail to the upper hinge / latch area a stiff box section for the door frame is defined.

Contrary to the design concept with a constant e.g. extruded or rolled frame profile the deep drawn inner panel can be designed with variable profile depths. Even here most OEM keep the frame constant in the directly visible area of a-pillar and roof rail and define the variable profile depth in the area of the b-pillar only.

In the front the door frame ends in the mirror triangle and in the rear the door frame ends in a blunt corner with the waist rail. Additional panels may reinforce the change over from frame and lower door case in the area of waist line.

The aerodynamic pressure distribution under driving conditions leads to a high vacuum in the joint area of a-pillar and roof rails. The load level can be increased by separate excitations of road and engine. The sealing gaskets and the corresponding surfaces on body or door frame must always overlap each other and only reduced relative deformation of the structures are expected. Otherwise loud wind noises in the area of the passengers head would occur when the air leaves the car through short term gaps.

The stiffness of the door frame and its connection to the door case must be defined in a way that preloads of the sealing gaskets, wind loads and other excitations are over compensated. Many OEM therefore over bent the doorframe in the joint area of a-pillar and roof rails. When the door glass lies on the door frame the door glass is over bent too. The disadvantage of this design is that the closing loads of the door and door glass is increased for the customer as the over bent door frame must be pressed back into its styling position when door is closed.

“Usually, complex dynamic loads affect the closures and therefore the doors of a vehicle body. Reducing this complex real time behaviour to the most important factors and defining appropriate load cases is usually a great challenge for any simulation engineer. The static linear simulation of the window frame stiffness is an example of such degradation. Here, the aerodynamic wind loads, as well as the road

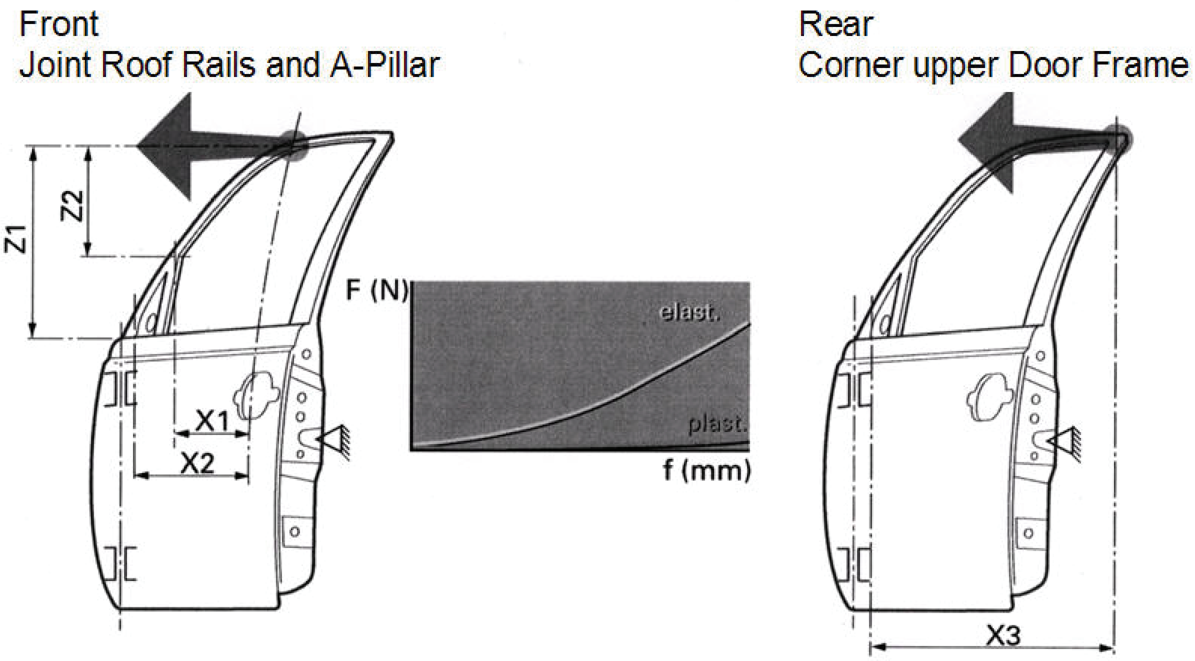


Fig. 5.33 Stiffness of door frame (Bekemeier, 1999)

and engine excitations, lead to complex vibration during the vehicle operation. These dynamic events can be reduced to a static load case, in which forces are applied to the front and rear window frames. From experience, the resulting stiffness required for satisfactory behaviour is known. The method allows fast and reliable identification of information needed for the designer to dimension the door structure (Lauterbach & Dick, 2007) (see figure 5.33). “

5.5.5 Waist Rail

The waist rail covers an important part of the stiffness of a door and safeguards the passive safety during frontal, rear and side impact (see figures 5.26 and 5.34).

Especially for cabriolets the waist rail defines an important load path for frontal impact. For the local reduction of gaps between doors and pillars in the load paths beads are designed or crash pads are mounted. The window regulator system (WRS) is pivoted to the inner waist rail to allow the improvement of sealing and guidance by adjustment of the door glass and door frame in Y direction by adjustment screws between WRS and inner panel in the sill area. Inner and outer seal are supported by the inner or outer waist rail to protect the inner door from water and dust. The stiffness of the outer waist rail must protect the door from forced opening through the waist line.

The door glass divides the waist rail into an inner and outer profile. The sophisticated design of the front and rear joints under consideration of door functions, producebility, stiffness and safety is complex. The inner panel and the one-piece inner waist rail panel easily define a closed profile because welding operations are mainly covered by the inner door trim. The outer waist rail must be defined e.g. by two separate panels as it is not possible to define a force fit between a single panel and the visible outer panel. So at the outer waist rail assembly order and definition of the contacts between rail panels, inner and outer door panels are a lot more demanding than inside.

The nonlinear dynamic simulation of the whole vehicle is ambitious and time consuming. It is the goal to optimize one of certain variants modelled in different ways to an optimum. So it is not useful to start with the simulation of the whole vehicle. The early layout of the body structures often is carried out under support of partial structures like the door inner panel combined with inner and outer waist rail. The geometry of these separate models can be easily defined, meshed and optimized. According to Lauterbach & Dick (2007) and Hänschke (2007) this approach reduces development time considerably and leads to reliable complete vehicle models. As there are no or only incomplete CAD geometries in the early phase of the product development CAE engineers model their partial structures themselves or use body structure libraries of former body structures or work with parametric structures defined in special CAE tools (see figure 5.18), (Hänschke, 2007; Zimmer & Schumacher, 2007).

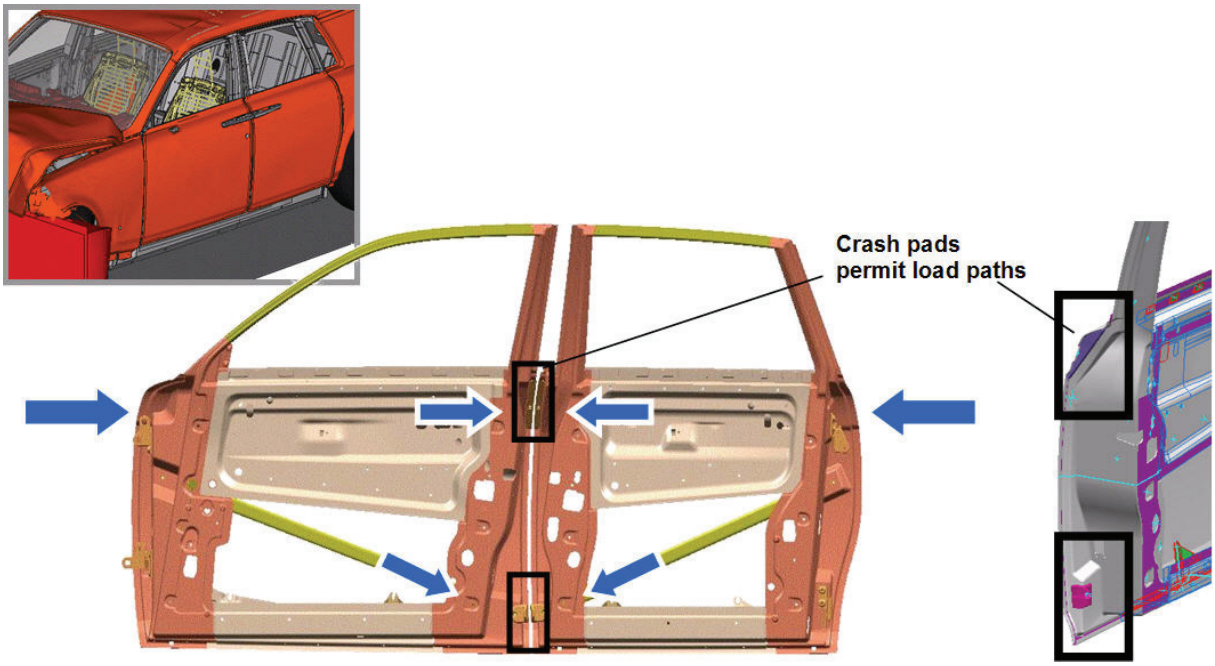


Fig. 5.34 Crash pads permit load paths through side doors (Lindermaier, 2006)

5.6 Concept Competition and Supplier Integration

“After mass production in the 1920s and "lean production" in the 1980s, the automobile industry is undergoing a new revolution. By 2015, automotive suppliers will have taken over large parts of R&D and production from the automobile manufacturers, achieving total growth of 70% in this process. During the same period, the auto makers will give up 10% of their current value creation, even though their output will increase by 35%. “(Wyman, 2004) (see figure 5.35)

The cooperation between OEM and suppliers has changed dramatically. The early integration of systems suppliers in the vehicle concept phase, distribution of responsibilities and partnership between all partners involved, concept competitions, target prices set by the OEM, focus of OEMs on core competences and related reduction of development depth for supplier components have defined a completely new qualification profile for suppliers during the last ten years.

Hundreds of OEM – supplier relations from concept development to delivery of components for mass production takes up a subordinate role in a vehicle development process. Besides OEM and systems suppliers the engineering suppliers take over most of the design work and a lot of simulation work. According to the master process (PEP) of the OEM they all follow the same goal and have to be coordinated from the technical and commercial stand points of the OEM. Basis for most technical communications and simulation processes are 3D CAD models which always have to be kept up to date and described precisely according to the requirements and maturity defined. Failures in sub processes endanger the overall goal (SOP) and lead sometimes to superheated catch-up games.

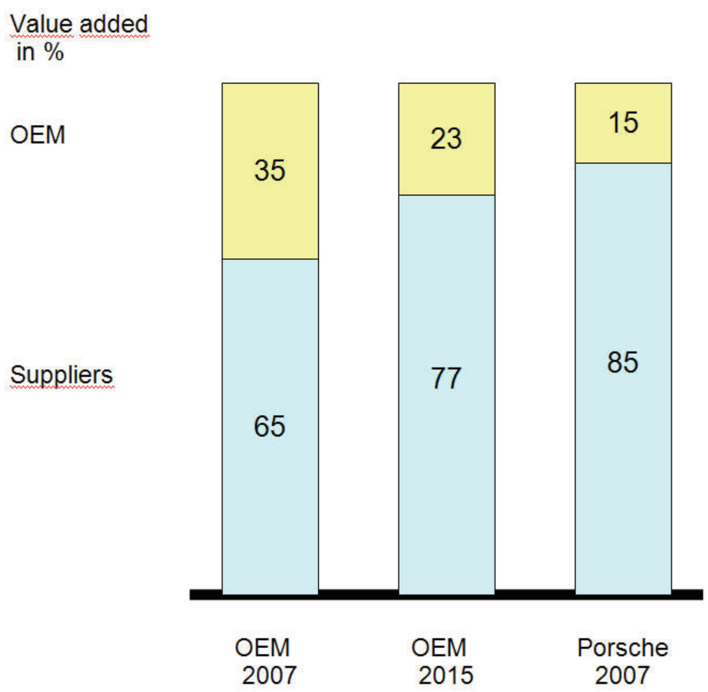


Fig. 5.35 Change of added value between OEMs and suppliers acc. to Wyman (2004) and Binder (2007)

For the promotion of the number of technical ideas, creativity and innovation OEMs define concept competitions at the start of a new project. In every competition several competitors are invited to apply their technical product proposals. Basis for the proposals are according to the enquired assembly a list of requirements defined by the OEM as well as package space restrictions and concept surfaces.

For the OEM this proceeding requires an extensive supplier management with assessment, development and risk management measures. The suppliers need to be integrated efficiently in suggestion gathering, creativity workshops, and concept competitions and at last in the concurrent engineering process.

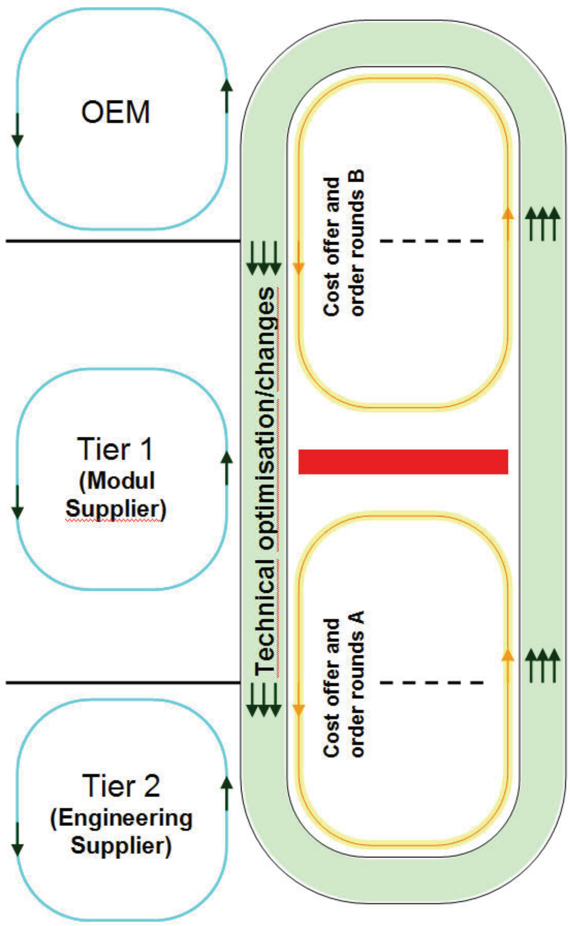


Fig. 5.36 Dynamic cooperation between OEM and suppliers

The technical cooperation between OEM and suppliers is a stringent dynamic process on a high level of outsourcing for engineering work. Otherwise the reduction of time in the development processes would not be possible. The commercial administrative process usually is neglected and often leads to depressive moods in technical cooperation. At the moment therefore OEMs and suppliers think about a restructuring of processes in form of strategic partnerships (see figure 5.36).

For systems supplier the reorientation of the OEMs leads to a new development strategy. While the former development of the systems suppliers was the realization of the product developed by the OEM, nowadays besides the actual system development innovation management, fundamental research and the independent pre development of sub and standard components belongs to their responsibility. The independent pre development may start 6 and more years before the component is produced in mass production. The development of project defined assemblies will be started with the concept competition of an OEM about 3 years before start of production. According to the complexity of the assembly about 6 to 12 month are needed to launch and guarantee a safeguarded mass production.

In context of concurrent engineering of vehicles and their necessary components an OEM expects from his suppliers at least the same competency and professionalism than the own R&D centre. Additionally a high level of communication ability is expected. These abilities not only include the CAD-, CAE- and data management systems but also up to date testing and analysing equipments as well as a high level of technical competence and performance. As long as the capacity of a systems supplier is insufficient, engineering suppliers are assigned to completely or partially develop components in a close coordination with the systems supplier and OEM.
Besides engineering suppliers the systems supplier engages sub suppliers for the development and delivery of sub components. Further suppliers of a systems supplier are companies specialized on production equipment and machines. These suppliers play an important role when the product is launched before SOP. The cooperation between all sub suppliers has to be carefully managed and synchronized by the systems supplier in cooperation with the OEM. The systems supplier takes over all the responsibility for the results of product development, product launch and delivery, and the absolute fulfilment of the list of requirements and target costs defined by the OEM.

5.7 Summary

This substantial chapter gives a small insight into the various development works of stylists, design, CAE engineers and related partners. Every unit of the automotive body will be developed with several variants with the objective of improvement towards one optimum. Every single development step of every variant of every unit needs the design of 3D geometry for evaluation.

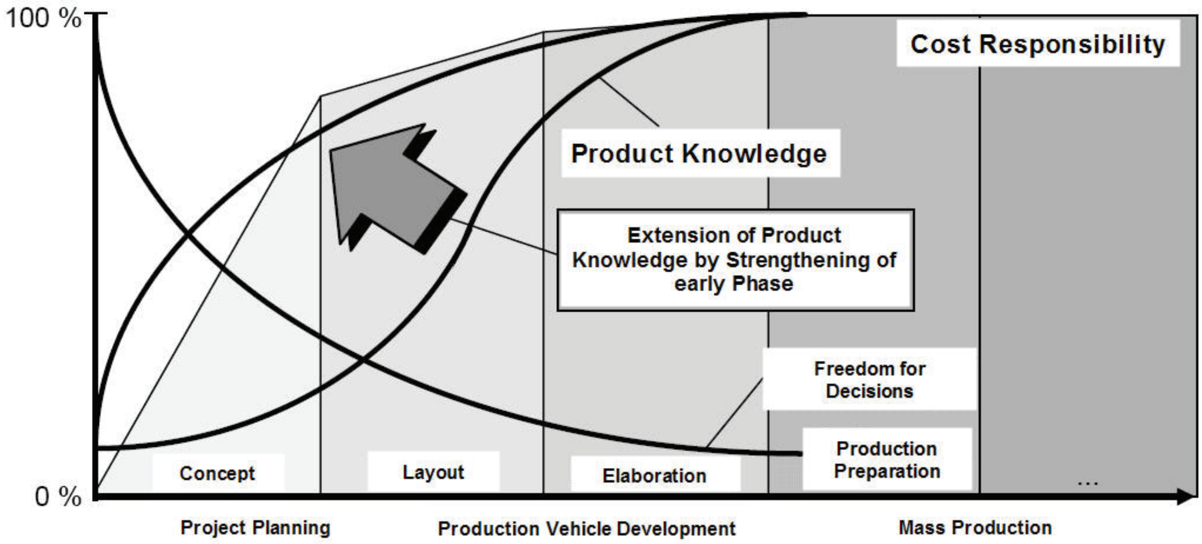


Fig. 5.37 Product knowledge and cost responsibility in PEP acc. to Wozny (1992)

It is the goal of this investigation to upgrade the product knowledge in the early phase of product development (see figure 5.37) by improvement and systematization of the design of 3D geometry under support of parametric associative design (PAD) and improvement of the distribution of design work within the concept phase.

In the following chapters the investigation will concentrate on the two central themes:

- Sections or profiles are the basis of surface and solid design. There for recommendations for the design and the application of profiles throughout different areas of automotive body design will be developed.

- For the distribution of design work in concept development it is useful to organise the design work of assembly components and their environment not according to the single parts of the assemblies but according to a zone based approach. Advantages and limits of zone based design will be investigated.

Chapter 6 Distribution of Design Work

6.1 Introduction

Distribution of work can be allocated according to detail parts, assemblies, modules or body zone location. The distribution depends on size, level of detail, the development phase and development time. During the early phase of product development engineers often design assemblies or modules with a low level of detail while in mass production (detailing) phase the design work is focused on single parts and is shared between many engineers. In the early phase of development several design variants and different approaches are investigated under support of the options within PAD. In the detailing phase key decisions are already concluded and fewer modifications are made which leads to lesser use of parametric approaches.

The early concept work on large assemblies i.e. closures can be distributed according to the single parts of the assembly and its mating geometries, according to design zones or a mixture of both. In the past layouts of large body assemblies were worked out on drawing tables and drawing boards. Car body designs were drawn in scale of 1:1, requiring drawing tables 7 metres long and 1.5 metres wide. Several engineers worked on one large layout e.g. side panels and side doors. The distribution of work was done according to their zone. One purpose of this research work is to apply the proven method of collaborative manual design work on large design tables to the methodical creation of PAD models hence permitting complete assemblies to be designed simultaneously in a zone-based approach.

For concept development according to detail parts approach independent of size or complexity, all the parts of an assembly are distributed between design engineers (see Figure 6.1). For reference all part specific parameters, Class-A surfaces, contact or distance surfaces of mating parts, joining points etc. are necessary. From this it follows that the design effort per part can vary a lot and every CAD-model is tailored especially for the individual part. Often design engineers of OEMs are forced to work according to this method because the corresponding parts of the assembly belong to different areas of responsibility such as BIW, closures, interior trim, passive safety, electrical systems etc.

For the zone based approach the automotive body is divided into several design zones (see Figure 6.1). With this approach design engineers share the design work for the side wall including closures, while they develop BIW, closures, corresponding surfaces of interior trim, control surfaces for passive safety, space for electric systems etc simultaneously. Firstly every design engineer designs the surface areas of all parts included in his zone. From this it follows that all design engineers must share and coordinate all references for the parts involved in the zones. It must be clearly identified which part functions control other part functions as well as positions and shapes of corresponding parts. The parameters and reference surfaces of all parts involved must be defined in every CAD-model of the mating zones as well as the transitions (e.g. planes) between the zones. Structuring and nomenclature used in the CAD-models must be organised. Once the single zones are completed trimming of parts and joints between single zones are designed as follow up. Then all zones must be brought together to derive the single parts from the zone based models. As a last step parts are detailed. For the zone based approach it is necessary to define project teams with designers from all disciplines.

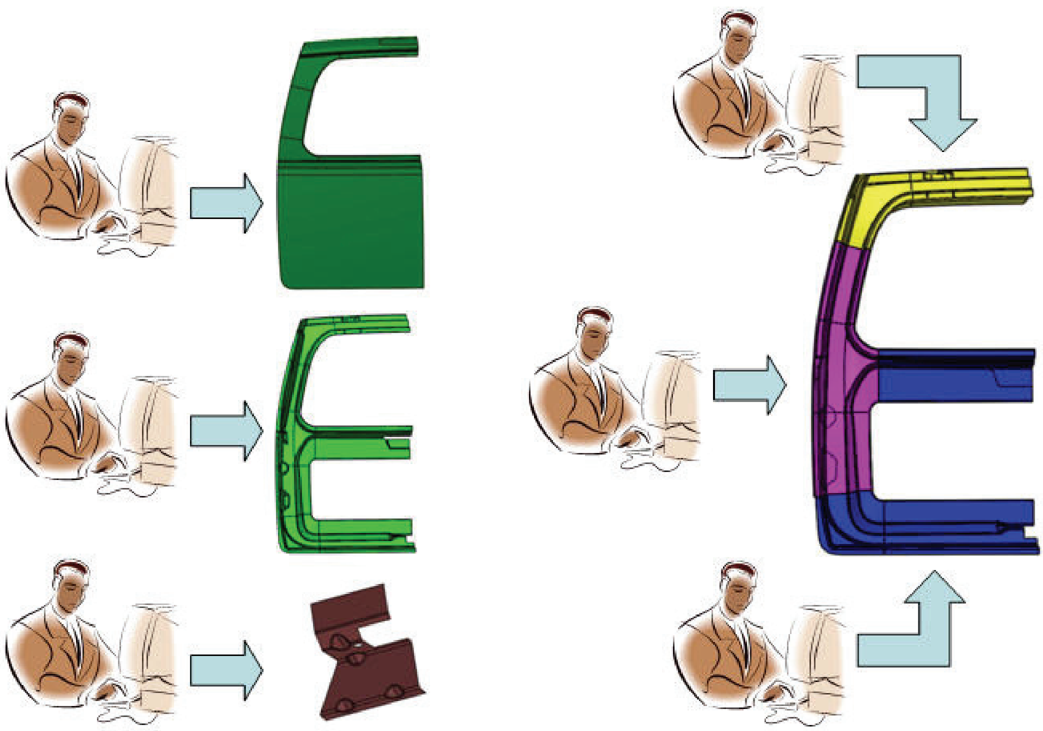


Fig. 6.1 Comparison of part based approach LHS and zone based approach RHS on the example of a tailgate (Aldag et al 2006)

Experience with several projects has shown that typical mechanical assemblies such as the sunroof system or the heating/air conditioning unit are more suitable for the part based approach while typical body parts like body structure, body panels, closures, exterior or interior trim panels are more suitable for the zone based approach. In this chapter the zone based approach is explained with its requirements, advantages and disadvantages.

6.2 Zone based Design in brief Examples

In 2004 the author use the zone based method on a side door design. The project was defined by the engineering supplier IVM, Munich and the OEM BMW and was carried out at the university (HAW Hamburg) with six students under the author's supervision. The results were poor due to the lack of experience with file based link management. A new project of a tailgate and its environmental geometries in cooperation with Volkswagen was carried out in 2006. Another project based on a side door was designed and investigated in cooperation with AUDI in 2007.

Volkswagen was interested to compare the Volkswagen part based approach with the zone based approach defined by the author. It was agreed a team of students would work out a tailgate and its mating geometries zone based from concept phase to detailing phase under the author's supervision. Volkswagen decided to share the design work in their parallel project in a part based approach from concept to detailing phase.

The results of the Volkswagen tailgate project are mentioned briefly in the introduction (Aldag et al, 2006; Reuter, 2007; Tecklenburg, 2006; Vickers et al, 2007):

The unequal complexity of the parts involved offer very different design requirements and resources. Every part oriented CAD model contains the design work from concept to detailing phase and is especially tailored for that part, modelling methods and details. Adapter models (containing all the references) and corresponding design models are closely constrained and therefore very unstable when it comes to larger changes and variations.

In the zone based approach the corresponding zones use a common adapter model which defines parameters and reference geometries as well as geometry for the transitions between the zones (for example construction planes). In addition to references used in zones the same definitions for the denomination of sub surfaces, attributes and publications for the common design of every single part surface have to be allocated. Construction models (kits) with designed zones showing different variants of the same body area can be combined to different variants of assemblies. Zone-based design models have a much higher degree of reusability than that of highly specialised part models. The side effect is that a team leader must be responsible for the entire model with all corresponding design zones. For the update sequence of the zone models involved in the project model and for all single parts that are included in the zones a good working group of different disciplines has to be established.

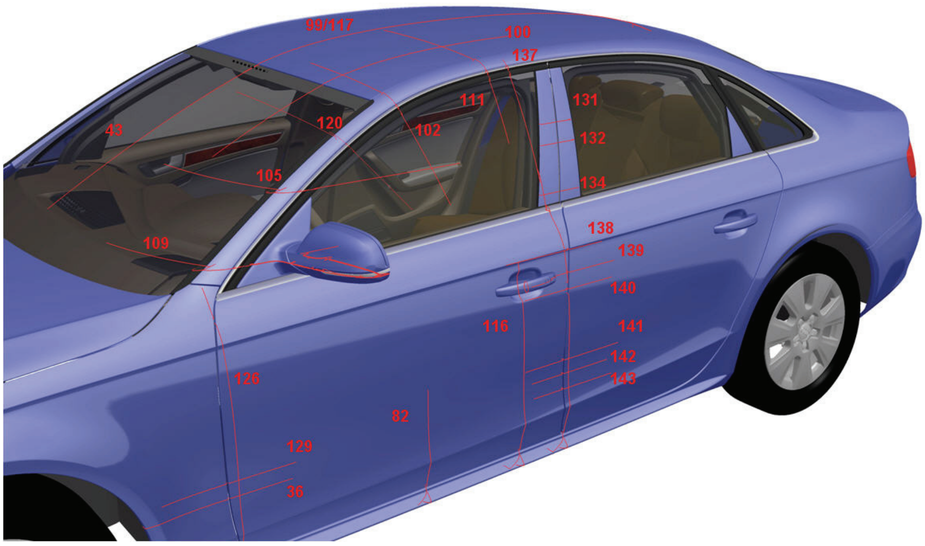


Fig. 6.2 Area of side doors with position of package-sections AUDI A8 (D4) (Nikol, AUDI)

Another brief example of two zones taken from the AUDI side door project explains more details about the zone-based design approach: Figure 6.2 shows the side of a passenger car. The complex functions of the side doors drive the design of the corresponding body structures.

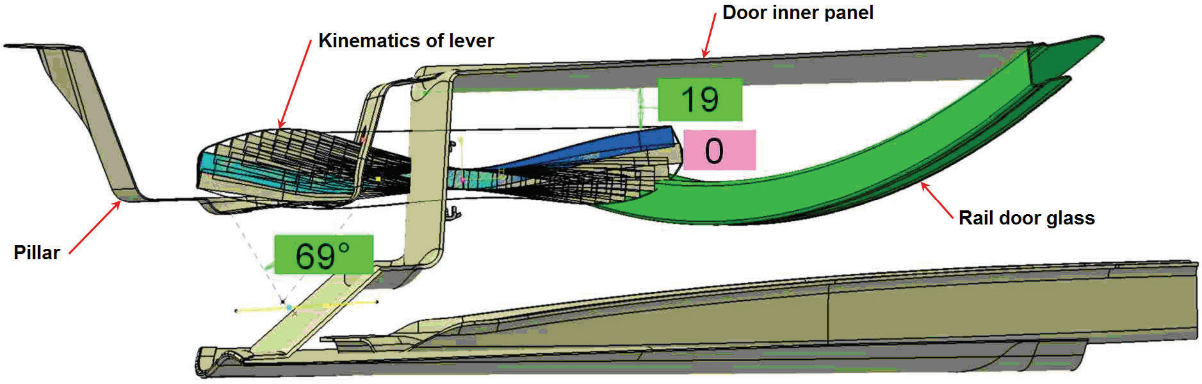
The zone of the upper A-pillar between waist and roof area includes sub zones of the circumferential glass bed of the front screen, the A-pillar and the front door. The distribution of design work can either be organised according to the single parts and sub assemblies or according to design zones of the front door and its corresponding parts (for example inner or outer door panel, inner or outer side wall etc.). In the concept phase of the design of a front door functional detail investigations are also needed which are integrated in the zone-based design directly or which are prepared in separate concept models. All parts have to be developed in context with corresponding parts (see list below).

For the example of a complete front door development the distribution of design work in a zone based approach could be organised in the following CAD models (Tecklenburg, G. (2008:1):

(I) layout body-in-white (BIW) door opening flange (flange door entrance between inner and outer side wall)
(II) layouts glass barrel[3], trimming door glass, window regulator
(III) lower A-pillar zone with layout of hinge axis and door gap
(IV) layout door check (see Figure 6.3)
(V) upper A-pillar zone
(VI) upper roof rail zone
(VII) lower B-pillar zone with integration of door latch and seat-belt retractor
(VIII) upper B-pillar zone with the inclusion of belt height adjustment
(IX) door belt and side mirror zone
(X) side impact carrier zone
(XI) door sill zone.

Due to time constraints the entire scope of the project could not be undertaken by a team of seven students during one semester. Thus, the projects had to be limited to the area below the waist line.

[3] Glass barrels are surfaces of revolution defined to fulfil the functions of door glass and window regulator as well as styling requirements. Trimming door glass means to cut out the shape of the door glass from the unbounded glass barrel surface.



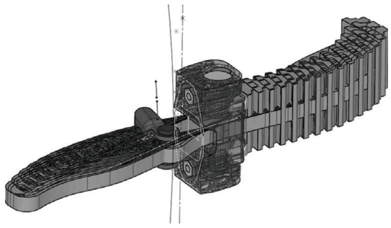


Fig. 6.3 Layout for kinematics of lever of the door check trap (lower picture Franzen, 2007)

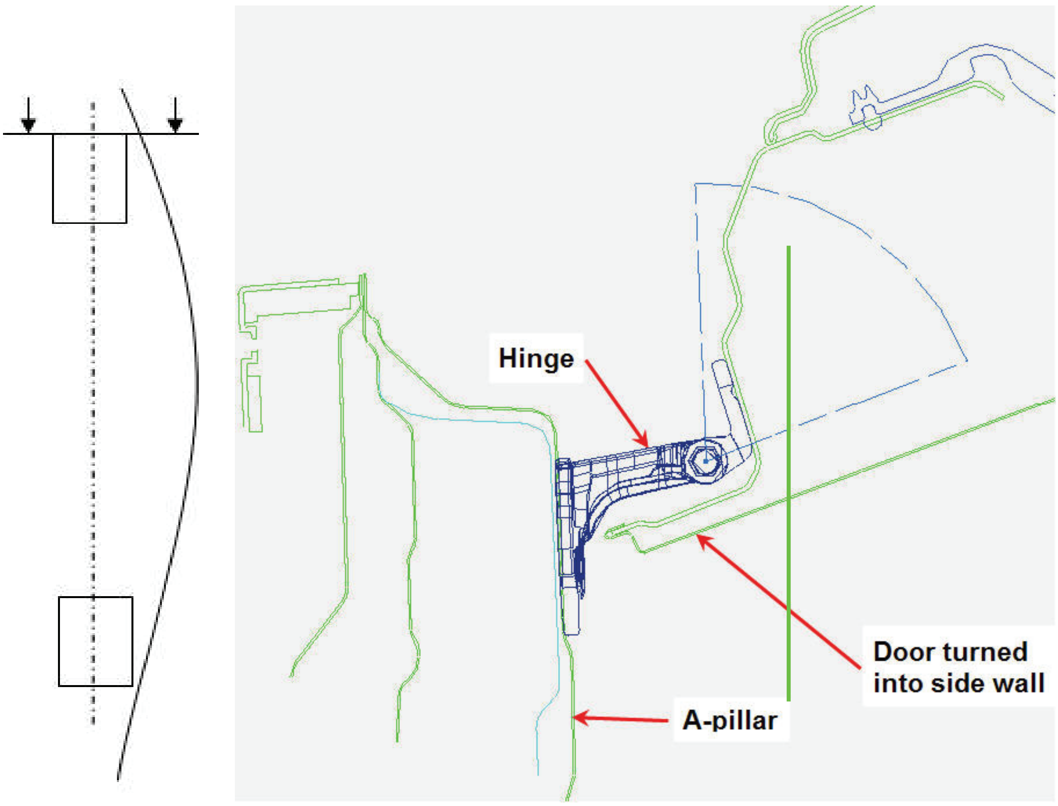


Fig. 6.4 Turned in front door with crash profile, hinge and lower A-pillar (Fischer, 2006)

The second example briefly describes the approach adopted for the development of a lower A-pillar zone (see Fig. 6.4) with hinge axis layout and door gap. In general side door hinges are arranged to turn the door tip into the side wall as turn-in arrangement. Dependent upon the current reference geometry (for example door gap and outer surface, A-pillar) an envelope for the positioning of the hinge axis must be designed (see workbench example Appendix 2). Then the door gap can be designed and on the basis of the front corner of the door and the outer surface of the A-pillar can be derived. The lower A-pillar zone shows three prismatic profile areas (design methodologies see Chapter 7):

(a) The door gap with first slope surface of the front wing and the crash-flange with its hemming flange of the door.
(b) The body side door seal flange of the BIW and its opposite seal surface on the door. Results of the BIW flange layout would be approved and updated in the zone model.
(c) The door's side door seal with its seal surface on the corresponding pillar is positioned as far outside as possible inside the hinge axis.

The inner and outer door frame in the hinge area (hinge frame) and the opposite hinge surfaces of the pillar are defined as planar surfaces relative to the hinge axis. The door function does not necessarily need an inclined inner and outer hinge frame but for tooling purposes the surfaces are inclined relative to tooling directions of the parts involved.

The mounting surfaces of the hinges are positioned in the directions of the main axes system. This allows an easy hinge adjustment for a flush outer panel and parallel door gap. The results of the separate layout of the door check are included in the zone model of the lower A-panel.

The following sections describe the distribution of work with zone based approach in more detail.

6.3 Zone based Approach for the Development of a Glass House

6.3.1 Introduction to the Project

In winter semester 2009 and part of summer semester 2010 in cooperation with AUDI, the engineering supplier GFI, both Neckarsulm, and HAW Hamburg the student team AUDI was instructed to design the body structure of the AUDI A8 (D4) (see Figure 6.2) in the area of the glass house in aluminium shell construction method (joining technique: punch riveting and in exceptional cases Metal Inert Gas (MIG) welding). Reference data were the body structure designed according AUDI-Space frame construction, the outer side frame including side doors, the inner trim panels and mounting parts such as curtain-airbags, handles, sun blinds etc.

Parallel to team AUDI the team GFI was instructed to design a new sunroof-system for the glass house of team AUDI. Team GFI used the same reference data. For further orientation they got package sections and exploded drawings of an old sunroof-system.

Both teams were asked to emulate the typical collaboration between an OEM and an engineering supplier and to update their concept development work regularly. Change scenarios for both teams were: a) a change of the Class-A shape of the roof, b) changes of sheet thicknesses, c) changes of airbag position and d) changes of shape and position of the opening line of the sunroof.

Because of the character of the assemblies team GFI was asked to distribute their concept design work according to the parts involved in the sunroof-system while team AUDI was asked to distribute the concept work in a zone based manner.

At the beginning of the project work a common product structure had to be defined. The contact zone where the supporting frame of the sunroof is riveted to the roof rails and the order of the overall assembly had to be discussed in detail. The first idea to rivet the supporting frame onto the roof rails side from underneath had to be rejected because of problems of space requirement for the water management in the corners of the sunroof. In the final order of assembly the supporting frame is riveted onto the roof rails from the top. The outer sidewall is applied in Y-direction and riveted to the roof rail and at least the roof is applied in Z-direction and laser-soldered to the outer side wall (see Figure 6.5).

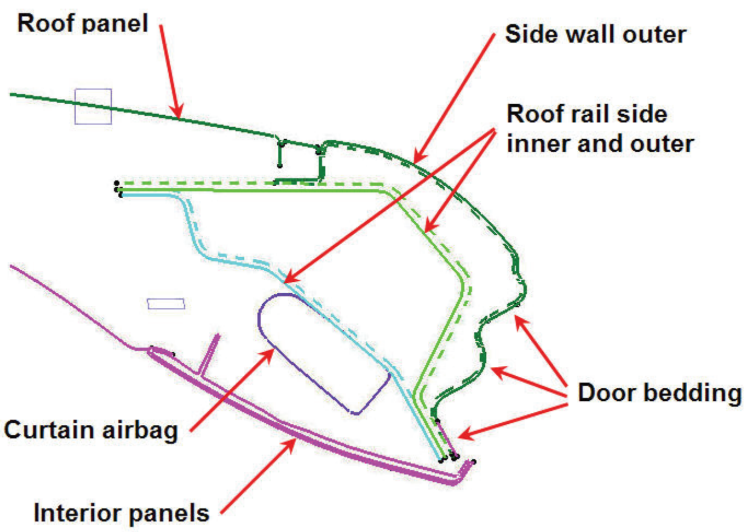


Fig. 6.5 Package section 111 - Roof rail side with curtain airbag, interior panels, sidewall outer and roof (compare with figure 6.2) (Bode et al, 2010)

6.3.2 Common Project Organisation of Teams AUDI and GFI

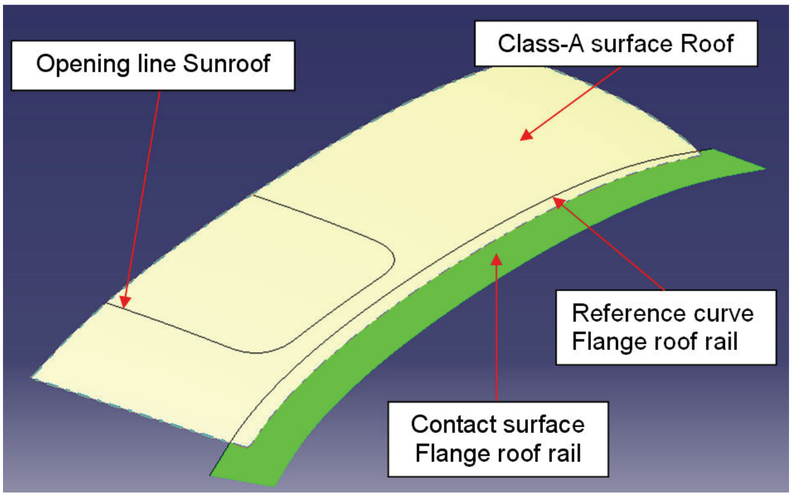


Fig. 6.6 Reference geometries for team GFI

As there is no Product Data Management (PDM) system available at Hamburg University of Applied Sciences (HAW) both teams were requested to define a file based project structure. The common project file contains the shared reference

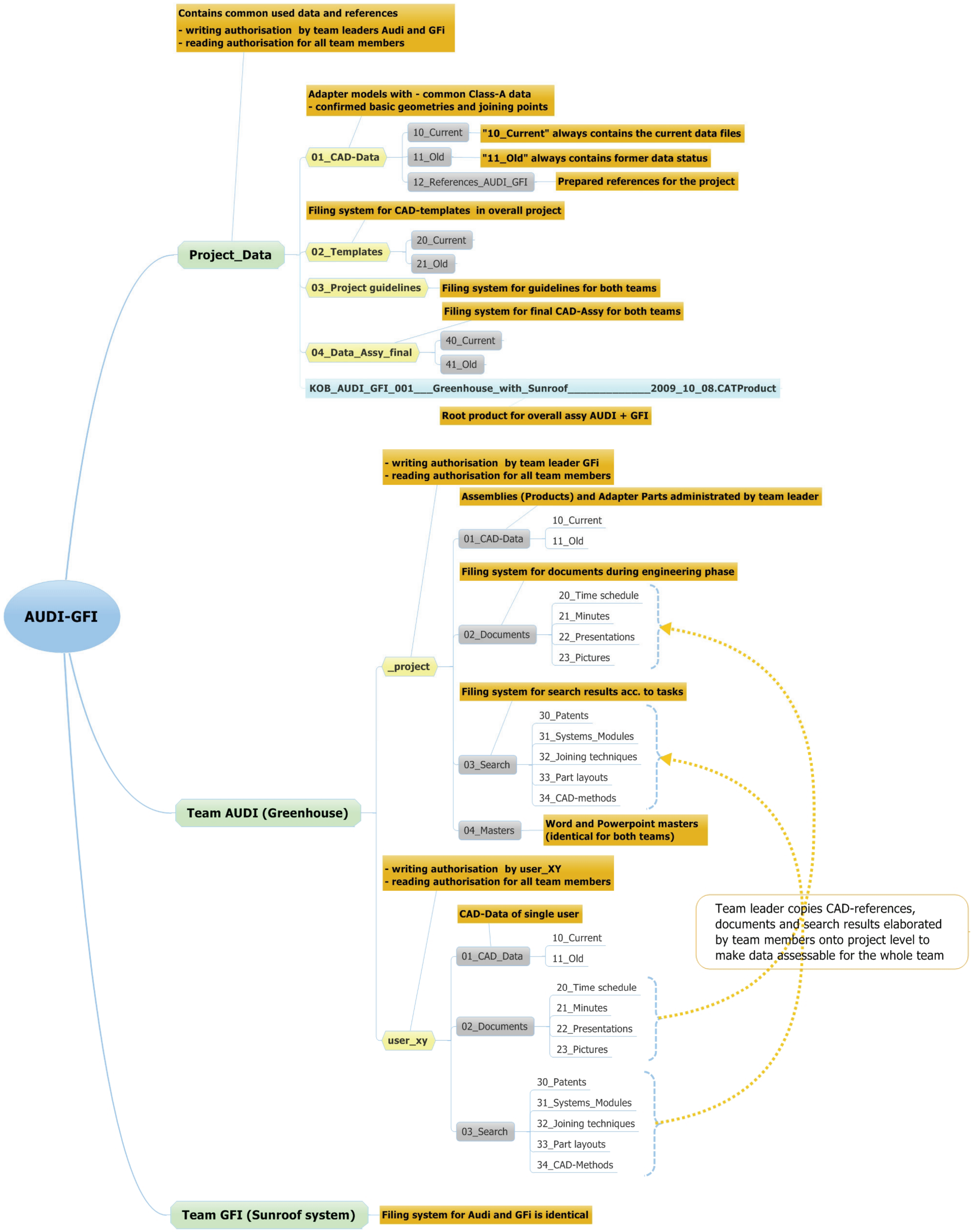


Fig. 6.7 File based project-structure for collaboration of teams AUDI and GFI acc. to Bode et al, 2010 and Untiedt et al, 2010

geometries such as package sections, Class-A surfaces, opening lines, contact surfaces and joining points (see Figure 6.6). In addition common design guidelines have been defined as well as a common CAD - start model (basic structure) and design templates. The final assembly (CAD-file) is the last but most important element of the common project file.

For every team, a team file with sub-files for the project and every single designer were defined. The respective team leader had writing and reading access for all files of the team while the normal team members had writing access only for their own file and reading access for all files of the team (see mind map Figure 6.7 and file in addendum).

Figure 6.8 shows the structure of the assembly and part models with main focus on team AUDI. In the assembly model (ASSY) of team AUDI two design phases for zone based and part design steps are defined. In the first phase the surfaces of all parts involved in the zone are designed as well as the transition zones (joints). In the second phase trimming of the parts is defined, surfaces are filleted and trimmed and parts are derived. Small additional parts such as stiffening plates are designed in this phase. In a further step of this phase parts are detailed (e.g. swages and recesses for mounting parts are added) and solid models are derived from the surface models. For FEM calculation mean (mid) surfaces of the structural parts and geometries for planned joining techniques are designed and placed in the structural tree.

It was agreed to use view links as well as manual links between all project phases. View links are automatically updated as soon as the reference in the adapter is exchanged or altered. View links inform the designers about changes of the reference geometries but are not used for the follow up design. Manual links are isolated copies of the reference geometries or view links used for the follow up design (explained in detail in chapter 4.2). As soon as the designer decides to take over the change he has to make an isolated copy of the view link, exchange the copy with the current design reference and update the design. For further detail information regarding the data management see mind map Figure 6.8 and file in addendum.

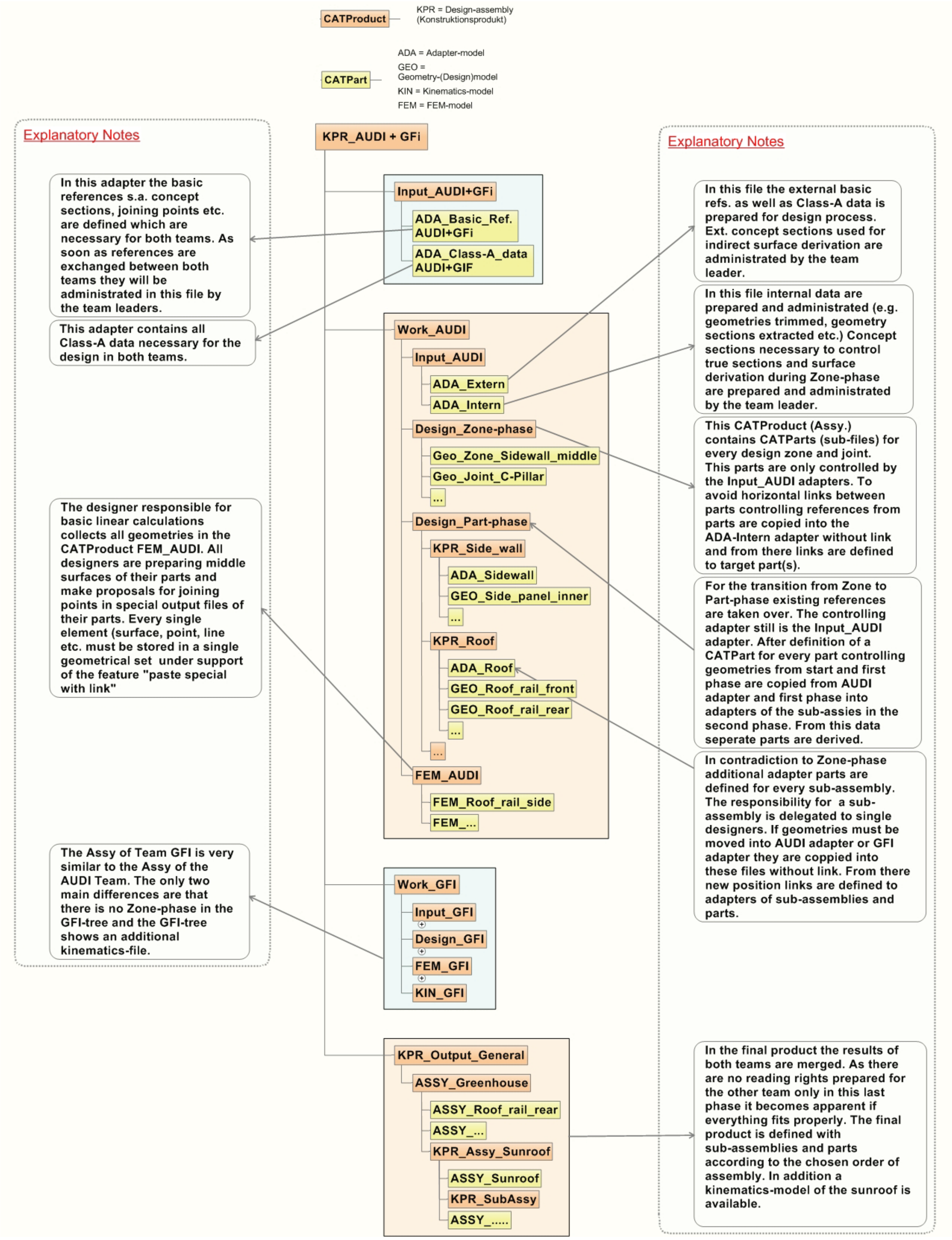


Fig. 6.8 CAD-model structure for collaboration of teams AUDI and GFI acc. to Bode et al, 2010 and Untiedt et al, 2010

Figure 6.9 shows the distribution of working zones and parts between five design engineers and one team leader of team AUDI. To get an understanding of the overall work Figure 6.10 shows the results of the part-phase of team AUDI.

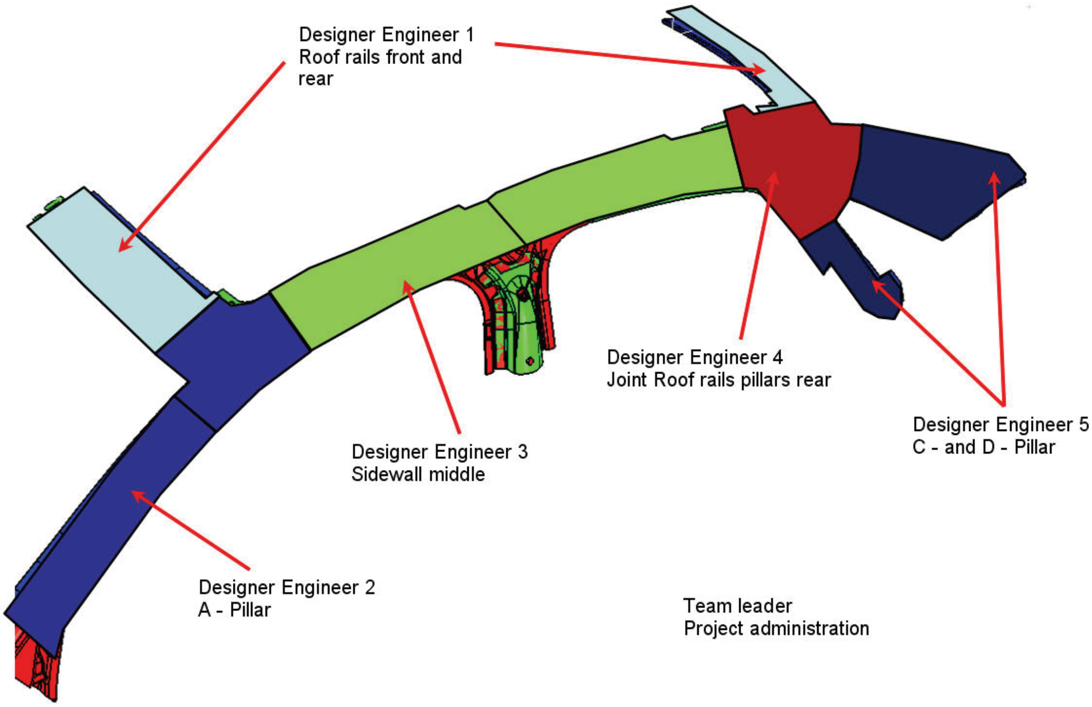


Fig. 6.9 Distribution of working zones and parts in team AUDI (Bode et al, 2010)

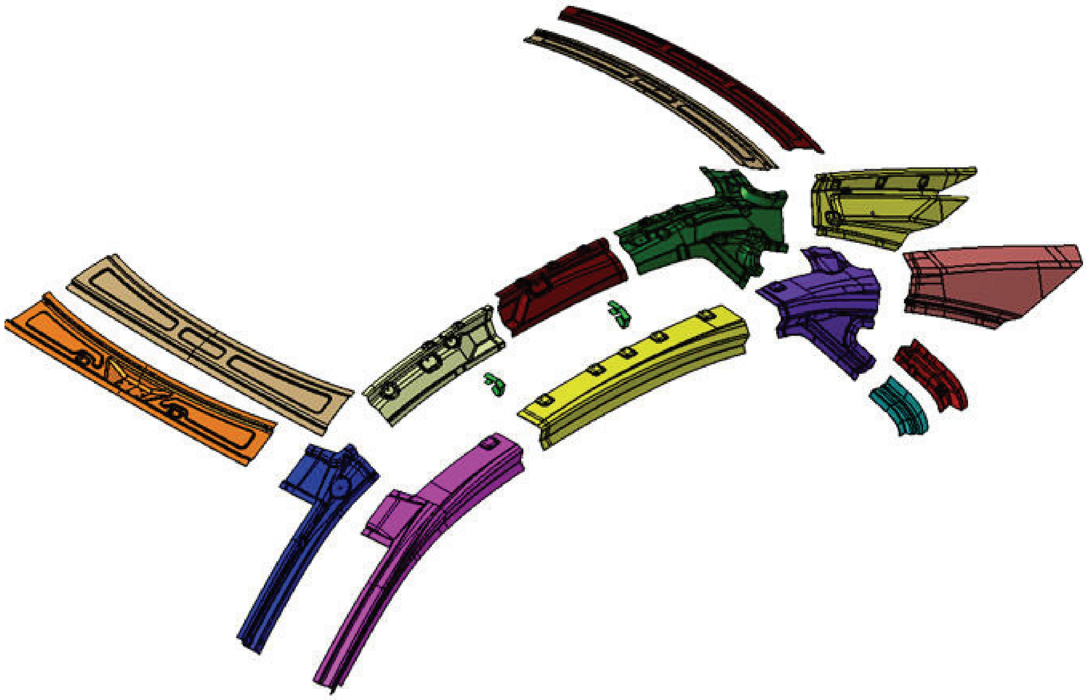


Fig. 6.10 Results of the part design at the end of the AUDI project (Bode et al, 2010) (Labels see figure 6.9)

6.3.3 Design Guide Lines

The execution of several team projects intensified the comprehension that a collaborative design work is only possible with clearly defined design guide lines. The guidelines developed and approved in the team projects for AUDI and GFI are presented in full detail in the addendum.

6.3.4 Start Model

The start model is a part defined with a hierarchical structure mandatory for all members of a design team. Only the consequent use of the start model combined with the project structure, the IDO-principle (Input, Design and Output - principle) and the guidelines lead to a continuous information flow and update safe behaviour of the CAD-models defined. Detailed information is presented in the addendum.

6.3.5 Manufacturing Requirements

It is the main challenge of product development to develop a successful and innovative new product for the international markets. In doing so new styling, new functions, fulfilment of future legal requirements and quality rules are essential. Manufacturing should produce high quality product but cost effective. Therefore well known manufacturing methodologies, controllable processes and production facilities are the goals of manufacturing strategy (Braess, Seiffert, 2007:1). Under support of PAD the requirements of both manufacturing and product development can be integrated in the early stages of concept development in material selection, product design and tolerancing towards the requirements of manufacturing (Brockmeyer, 2010).

In concept development production methodologies should already be taken into account, e.g. design of parts and assemblies under consideration of part size, tooling direction and sequence of assembly, position and style of fixation points for mounting and measurement, width of mating flanges and accessibility of flanges by weld or rivet guns or other devices. The start model defined for team AUDI specifies the use of tooling direction and release angles. Other topics relevant for manufacturing such as sequence of assembly, width of flanges, distance of weld spots etc. must be determined early.

Besides the demands mentioned team AUDI had to take into account two different rivet guns for self piercing rivets delivered by AUDI in form of 2D silhouette and 3D skin CAD geometries (see Figure 6.11). Accessibility of flanges and unobstructed operation of the rivet devices had to be checked within the assembly development at any time. For the two different rivet guns in 2D and 3D four power copies (see glossary) were defined. Input-geometries are the flange surface, flange corner and the rivet point defined. The rivet direction (from thin sheet towards thick sheet) and the rotation angle around the normal axis are adjustable under support of parameters. The segmented envelope of the rotated silhouette and a clever definition of extreme points for contact allow the control of accessibility and guarantee the clearance between the rivet gun bracket and the flange in a certain angular range. By these power copies demands of manufacturing could be approved at an early phase of concept development and after completion of part phase. Nevertheless these proceedings must be approved by manufacturing under consideration of further demands at a later stage.

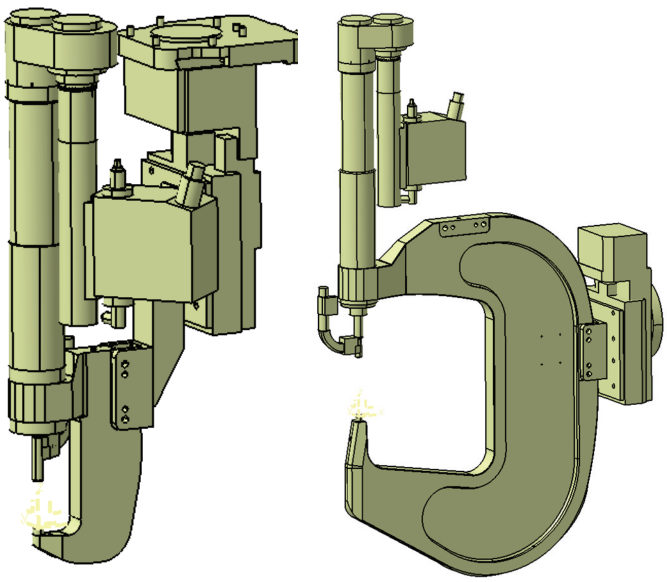


Fig. 6.11 Rivet guns with different bracket sizes (mouth widths) (Bode et al, 2010)

6.3.6 Process Chain from Concept Section to derived Parts

6.3.6.1 Zone-Phase

The zones shared by the five designers are divided by transition planes. On these planes shared concept sections are defined. These true most sections called Master_Sketches in the start model show the whole cross section. Often the concept section is not true and can not be used for surface derivation. The main purpose of the concept section is to show the complete constructed space and to control the true sections of prismatic profile areas. The process chain of sections is linked up in a way that all definitions of segments and parameters are made in the (radial) concept section but the surfaces are derived from the controlled true (compound radial) sections. For the update safe design of fillets in the part- phase the surfaces are designed with excess width. A detailed description of the process chain from package section via true (radial) section to true (compound radial) sections is presented in the next chapter.

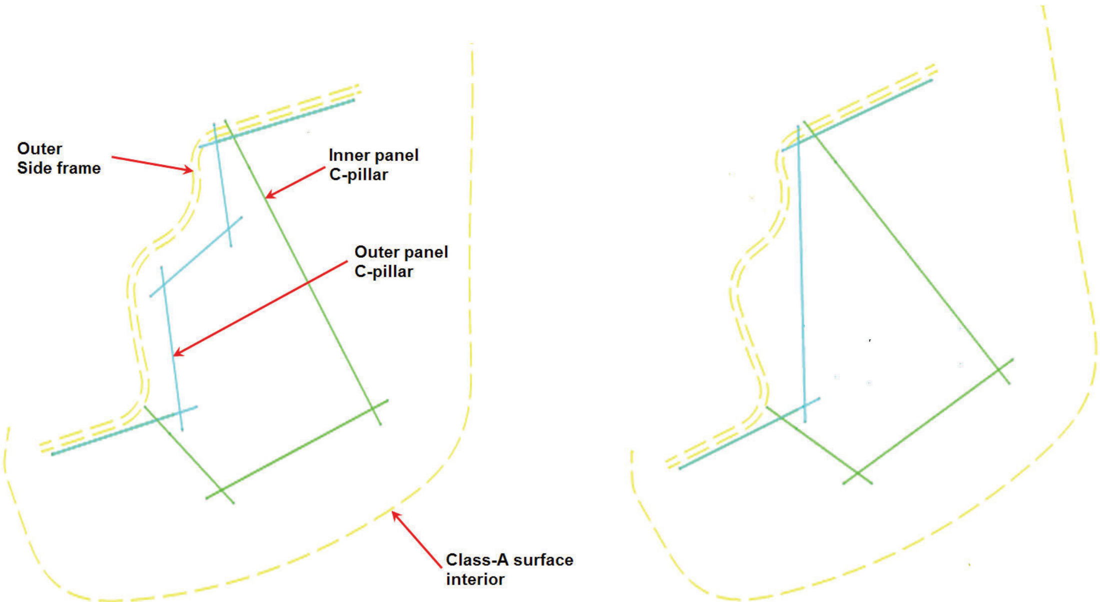


Fig. 6.12 Concept section 1 C-pillar LHS and 2 RHS (acc to Bode et al, 2010)

For the consistent cooperation between the designers of the team and mating teams it is necessary to clearly define who is responsible for controlling geometries. The person responsible develops the shared concept section and hands over the surface bands and its spine and guide curves derived from the design process to the other dependent designers. After discussions of the boundary conditions the teams made the following decisions: roof rail side controls the A-pillar, roof rail front controls the joint integrated into the A-pillar zone, the B-pillar controls the joint integrated in roof rail side, the roof rail side controls the supporting frame of the sunroof, and roof rails side and rear together with C- and D-pillar control the joint side wall rear.

On the basis of guidelines such as sequence and direction of assembly or minimum distance towards Class-A surface interior first concept sections of the C-pillar are designed on a plane perpendicular to the spine and guide curve flange BIW outer side frame. As both spine and guide curves of flange BIW outer side frame and opening line side window differ less than 3° from a parallel direction surface bands are directly derived from the concept section and its spine and guide curve because the break down into true sections is not necessary from the geometrical stand point. The inner surfaces (punch side) of the pillar are designed (see Figure 6.12).

Alternative to concept section 1 a second concept section of the C-pillar is developed which shows a simpler outer panel because the panel does not follow the steps of the door bedding defined on the outer side frame. To compare both variants a linear calculation is carried out in CATIA (see Figure 6.13). For the load case chosen 1mm sheet thickness, a force of 100N on the upper corner, material aluminium and realistic restraint conditions are defined. As expected the distribution of tension is better in the multistage panel. As additional corners will increase buckling stiffness the concept section 1 is chosen for the further design.

The design approach for the D-pillar is more complex than for the C-pillar. At first a true most concept section is designed on the transition plane to joint side wall rear perpendicular to the theoretical intersection curve of rear and side walls (see Figure 6.14). Two true local sections for the prismatic profile areas of window beddings side and rear windows are designed and linked to the concept section. From the true sections surface bands are derived under support of associated spine and guide curve. Between the two prismatic profile areas warped surfaces are designed under consideration of Class-A surfaces exterior and interior. In addition for the inner panel the contact surface of the curtain airbag[4] for the rear passengers had to be taken into account.

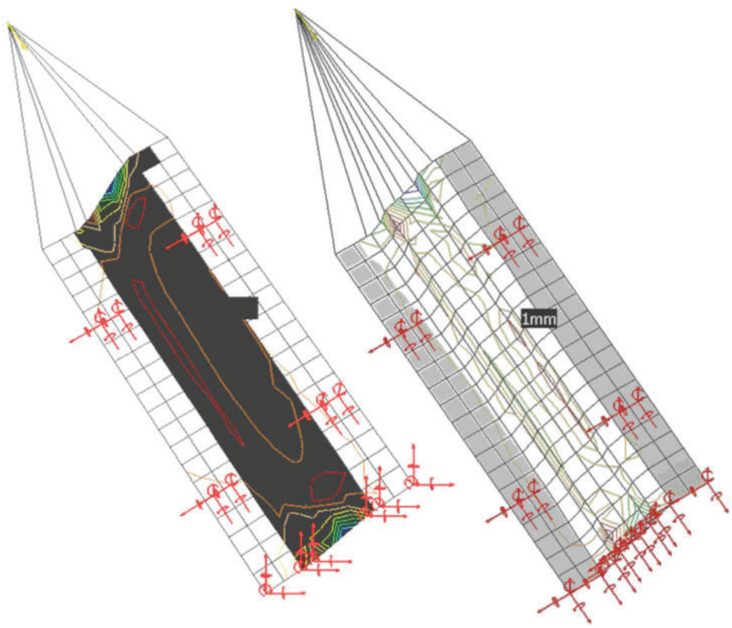


Fig. 6.13 Comparison of two different outer panels C-pillar (Bode et al, 2010)

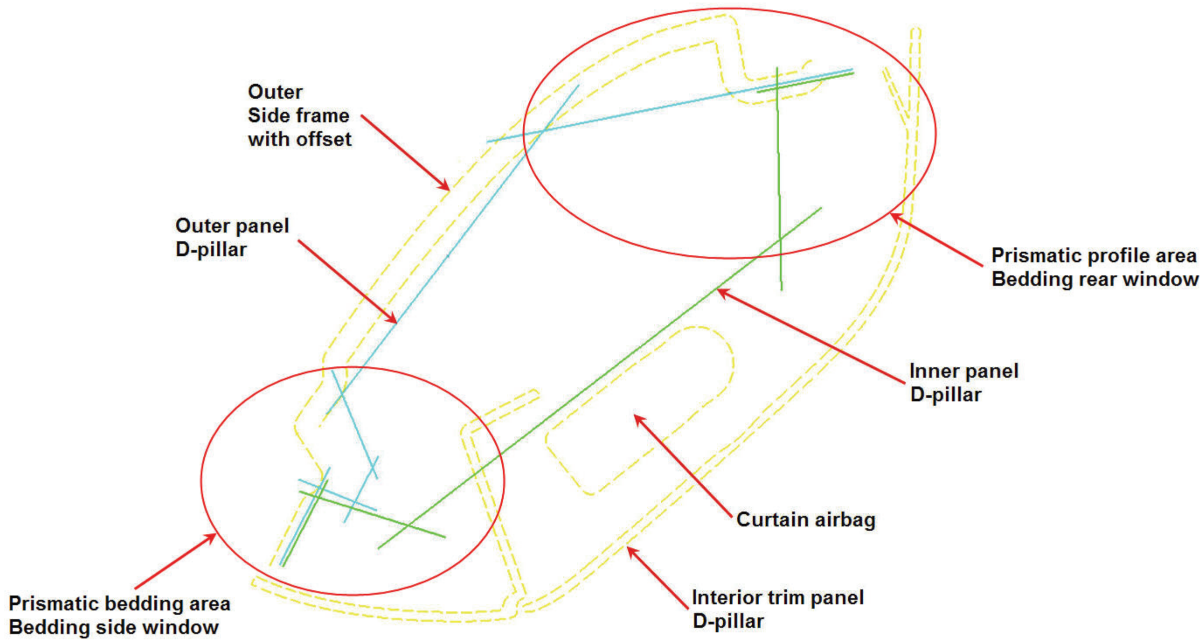


Fig. 6.14 Concept section D-pillar (acc to Bode et al, 2010)

[4] Curtain airbag is an airbag system mounted to the pillars or roof rail side. In case of an accident the airbags define a cushin system hanging down inside the side windows to protect the heads of the passengers.

For the design of joint side wall rear it was decided to control the surfaces of the joint by all it's mating profile zones. The references for the joint are the external references of Class-A surfaces interior and exterior and the transition planes as well as the internal references of all concept surfaces and associated spine and guide curves of the mating zones (see Figure 6.15).

In zone-phase the design of the joint side wall rear takes place in file Sheet_Metal_Part under support of the input data mentioned (see Table A.4.1). Most of the design steps are performed in file Part_Geometry_Untrimmed which is structured further according to the IDO (Input, Design and Output) principle. The file Part_Geometry_Trimmed_Non_Pierced is only used to prepare the output data. In the input sub file of file Part_Geometry_Untrimmed the Class-A surface of side wall outer and the new transition planes for the two sheet metal shells of the rear roof rail as well as all surface bands and associated spine and guide curves of the mating zones are stored. All surfaces are checked according to their designed side and material vectors and trimmed by the bounding box. In addition the input file contains the intersection curves between the transition planes and corresponding surface bands (see Figure 6.16).

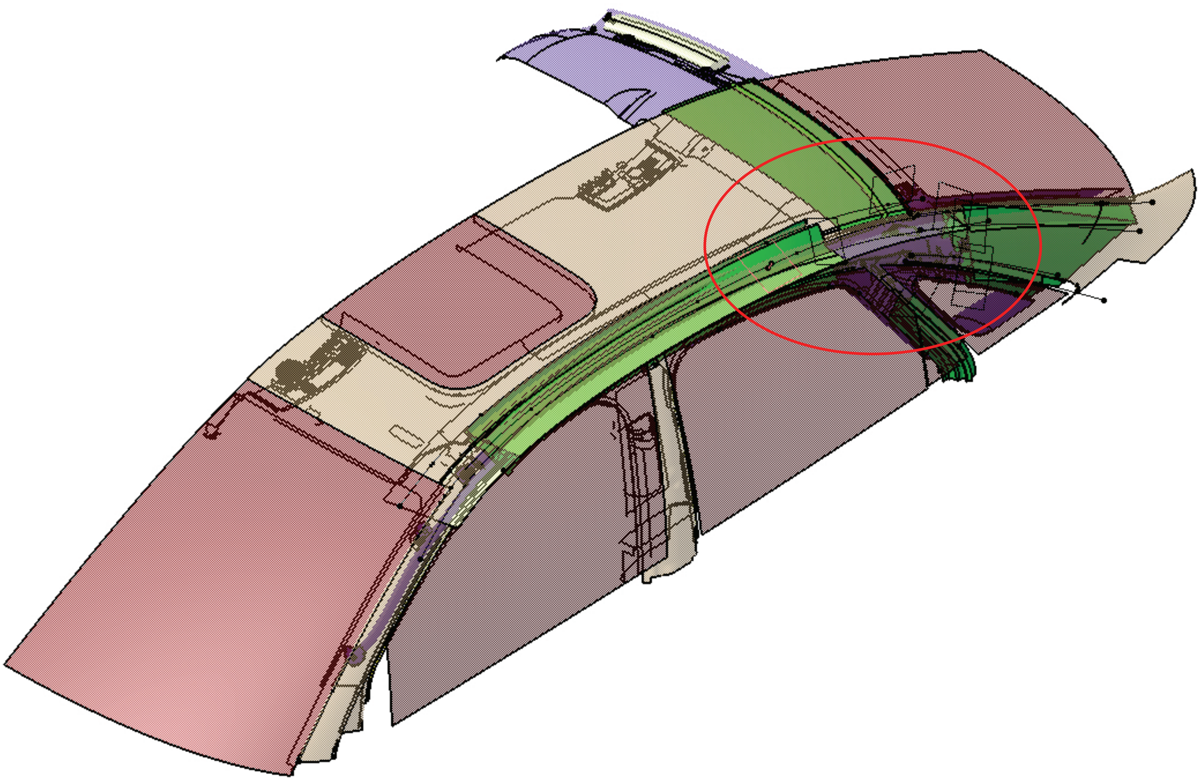


Fig. 6.15 References for the zone-phase of rear joint sidewall upper (Bode et al, 2010)

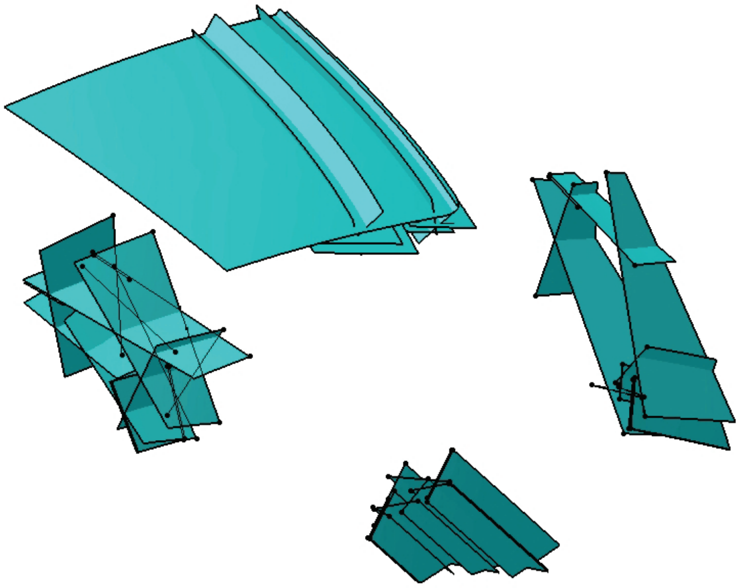


Fig. 6.16 Prepared Input-data for design of rear joint sidewall upper (Bode et al, 2010)

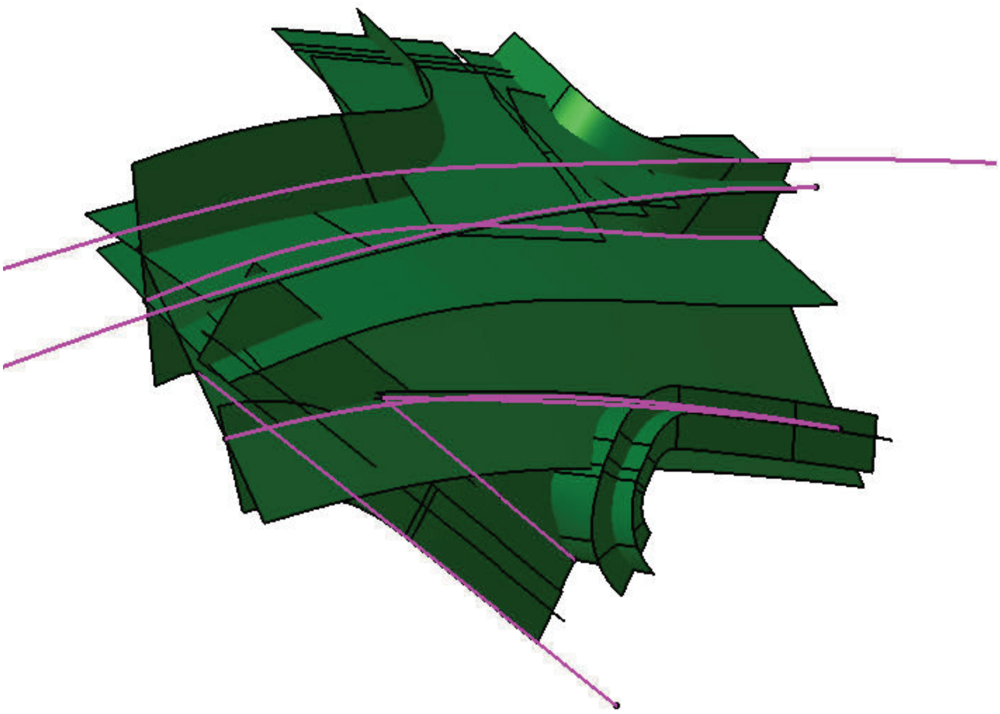


Fig. 6.17 Concept geometries inner shell rear joint sidewall upper (Bode et al, 2010)

The design sub file is structured further according to the outer and inner shell of the joint. These sub files are structured further according to the single transitions of the mating parts. At first spine and guide curves for concept surfaces of the joint are designed. Afterwards continuous transition surfaces are derived from these curves and the input geometries (see Figure 6.17). Under support of bounding surfaces designed for the areas of roof and rear window concept surfaces of the joint are trimmed evenly.

The output sub file contains all “Invert”-geometries of the isolated concept surfaces and curves. The “Invert”-geometries are published for the use of the data in the part-phase.

6.3.6.2 Part-Phase

In the part-phase trimming geometries for the parts are definitely defined and all parts are derived from the concept geometries of the zone-phase. Parts are detailed e.g. with recesses, stiffening swages and holes. Small additional parts such as stiffening plates are designed in this phase. Designers now are responsible for the parts of their zone, but it is possible that large parts are extended over several zones.

Generally an adapter is placed at the head of the structural tree of part-phase to collect all input-geometries necessary. In this adapter all “Invert”-geometries of the isolated concept geometries would be stored. Team AUDI decided to define one model structure for zone- and part phases so that the model structure of the part phase has no phase adapter and can not be opened separately. All individual geometries necessary for one part are copied directly from the output-data of the zone phase into the adapter of the individual parts.

On the example of inner panel rear joint sidewall upper the design work in the part phase is described. Figure 6.18 shows the completed parts of inner and outer panels of rear joint sidewall upper. Besides the concept surfaces of the joint developed in the zone phase concept surfaces from the mating zones roof rail side and C-pillar are necessary for the part design (see Figure 6.19).

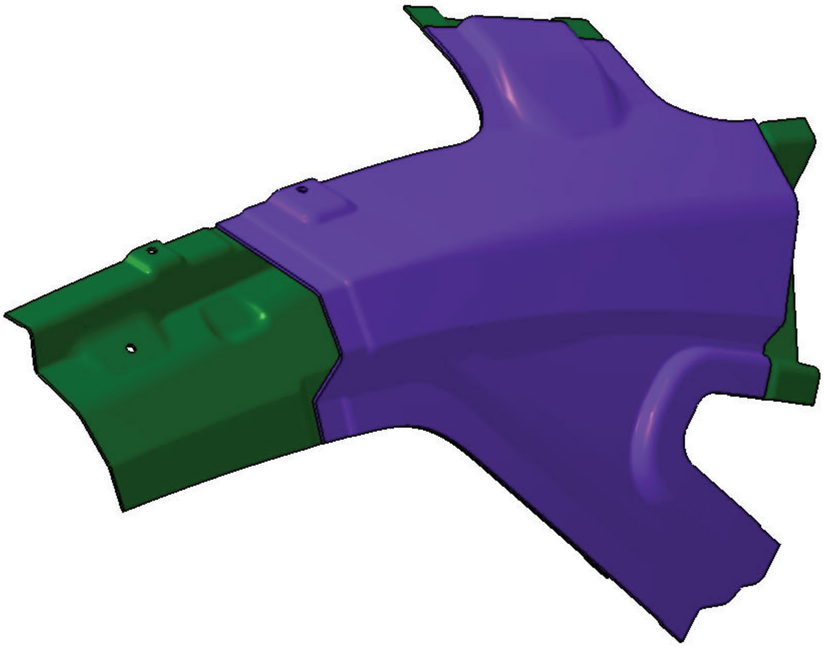


Fig. 6.18 Completed parts inner (green) and outer (blue) panels of rear joint sidewall upper (Bode et al, 2010)

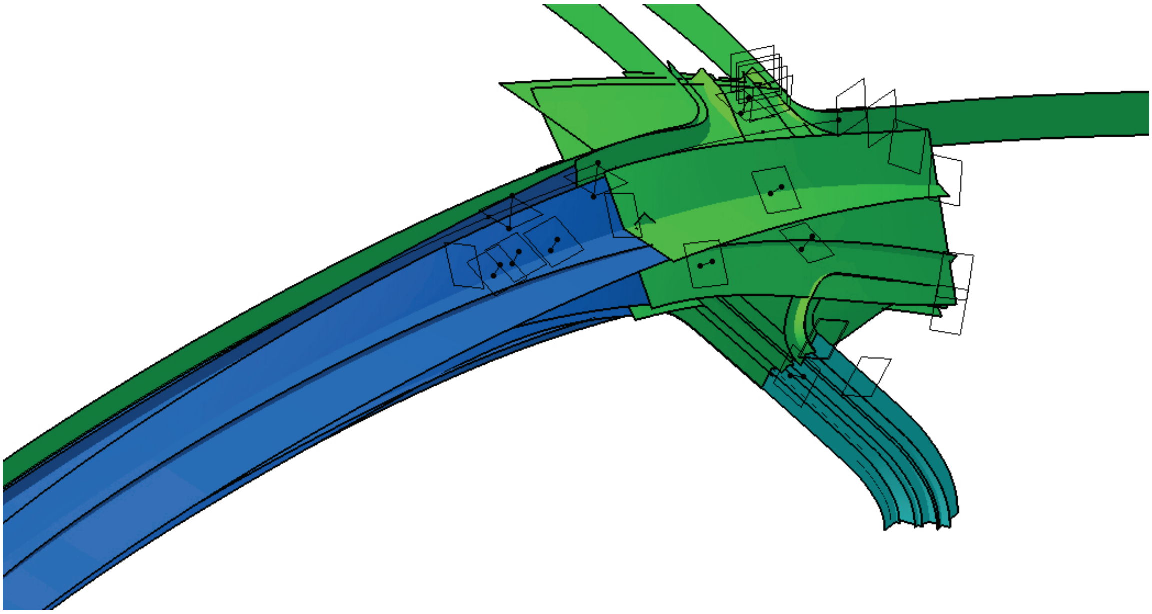


Fig. 6.19 Input data for part-phase of joint sidewall upper (Bode et al, 2010)

The design of the part inner panel joint sidewall upper is performed in the file “Sheet_Metal_Part”. According to the guidelines defined on the basis of the design methodologies of the OEM´s start model first off all the basic surface assembly with untrimmed outer edges is designed in file “Part_Geometry_Untrimmed” (see figure 6.20). The example of the opened file “Inner_flange”makes clear that several design steps are necessary to supplement and assemble the primary surfaces of the zone-phase for one subassembly of surfaces.

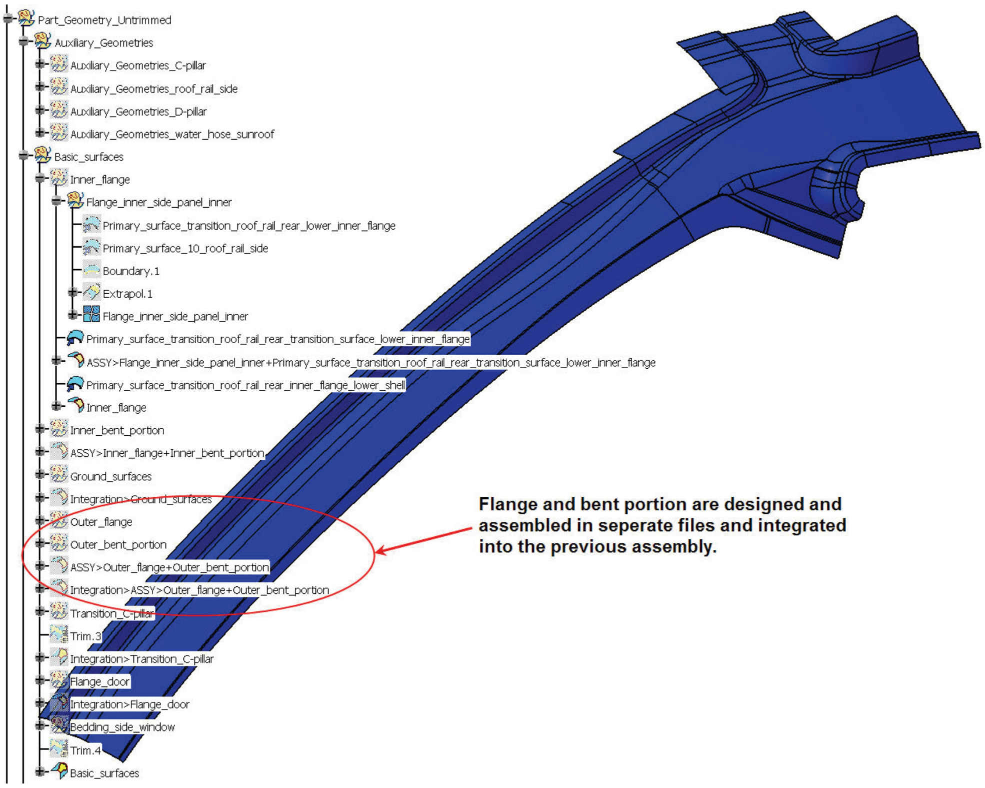


Fig. 6.20 Basic surface assembly of inner panel rear joint sidewall upper (acc. to Bode et al, 2010)

This methodology specifies that always again a new subsurface or a subassembly of two new sub surfaces is designed and integrated into the previous assembly under support of features like “Join” or “Fillet”. Because of the high number of subassemblies the design work in the part-phase is substantial. In doing so the basic guideline to keep the structural tree traceable and legible stands in the foreground.

After the basic surface assembly is completed the detailing work starts with the design of recessed surfaces for the connection with the C-pillar and is proceeded with the design of several recesses (e.g. for the fixation of the supporting frame of the sun roof or the deformation elements of the handhold roof rail) (see figure 6.21).

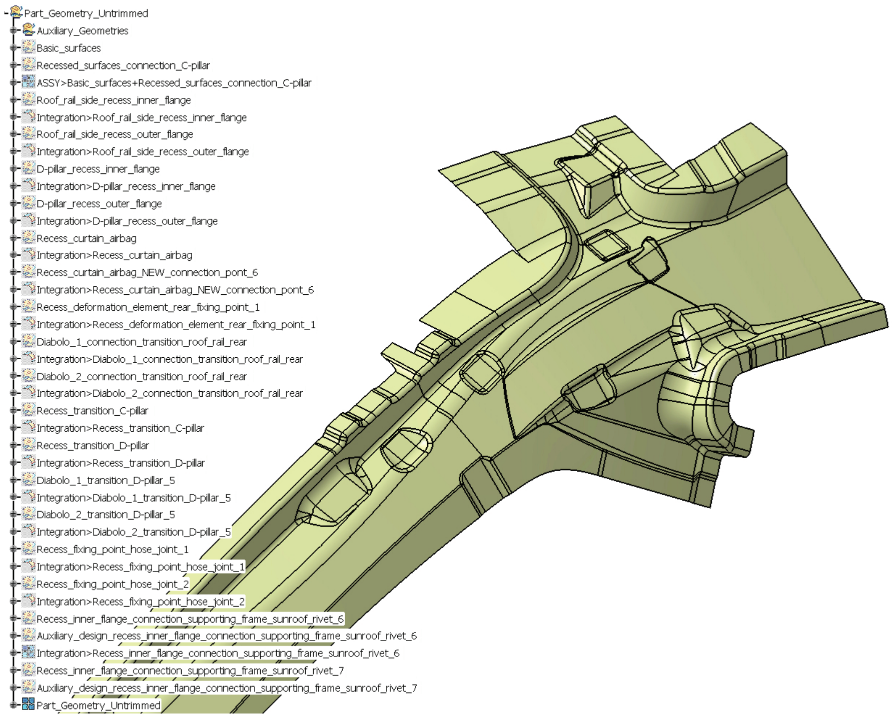


Fig. 6.21 “Part_Geometry_Untrimmed” of inner panel rear joint sidewall upper (acc. to Bode et al, 2010)

After completion of eighteen recesses in file “Part_Geometry_Untrimmed” the part is trimmed under support of trimming geometries defined in the zone-phase for the roof and rear window area as well as planes on the transitions to C- and D-pillars and roof rails side and rear. The file “Outer_Trim_Contour” contains all the geometries necessary for trimming of the outer edges. The trimmed part is stored in file “Part_Geometry_Trimmed_Non_Pierced”. Under support of positive geometries for every hole (extruded surfaces) the necessary holes are designed. The file “Hole_Feature” contains all extruded surfaces to be subtracted from the non pierced part to design the holes. In file “Part_Geometry” the completed surface model (punch side) of the inner panel is designed and stored (see Figure 6.22).

To conclude the design work of the part all contact surfaces to mating parts (e.g. fastener water hose and supporting frame of sun roof) are stored in form of linked and isolated geometries. “Invert”-geometries of the isolated geometries are defined and published for transfer to other designers. In the file “Output_FEM“ all relevant data for FEM calculations are defined such as the mean surface of the part, the seventeen rivet points and the glue line for the connection with roof rail rear. These geometries are exclusively stored in a linked format and published for the further use of other designers. Under support of “Thick surface”-feature a solid model of the part is defined and added to Part Body (final part) in the structural tree by a Boolean operation (see Figure 6.23). To complete the part the material is defined in the Part Body, material and its density are defined and volume and weight of the part are calculated in the set of explicit parameters. All other large parts are designed and structured accordingly.

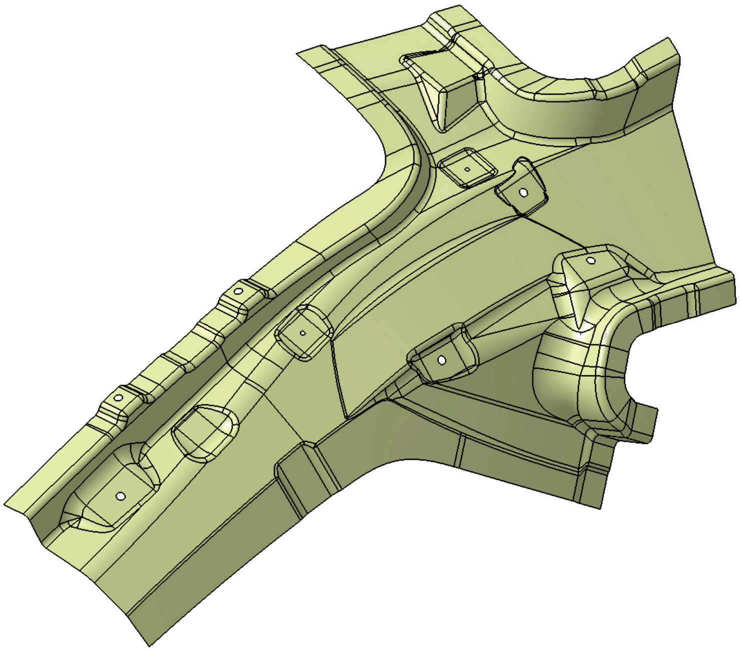


Fig. 6.22 Completed surface model inner panel rear joint sidewall upper (acc. to Bode et al, 2010)

6.3.6.3 Assembly Glass House with Sun Roof

In the part-phase the parts and subassemblies of team Audi are structured zone-based using an adapter for every zone (designer). For the total assembly of glass house with sun roof a separate CAD-model is built showing the parts and subassemblies according to the sequences of assembly. The separate CAD-model administrated in the common project file is necessary because of the different reading access of both teams. The separate CAD-model can be read by all team members of both teams. In the total assembly the manufacturing subassemblies ASSY Body in white, ASSY Sun roof module, ASSY Mounting parts exterior and ASSY Mounting parts interior are defined. Parts and subassemblies designed by the team members as well as Carry Over Parts (COP) are structured in context.

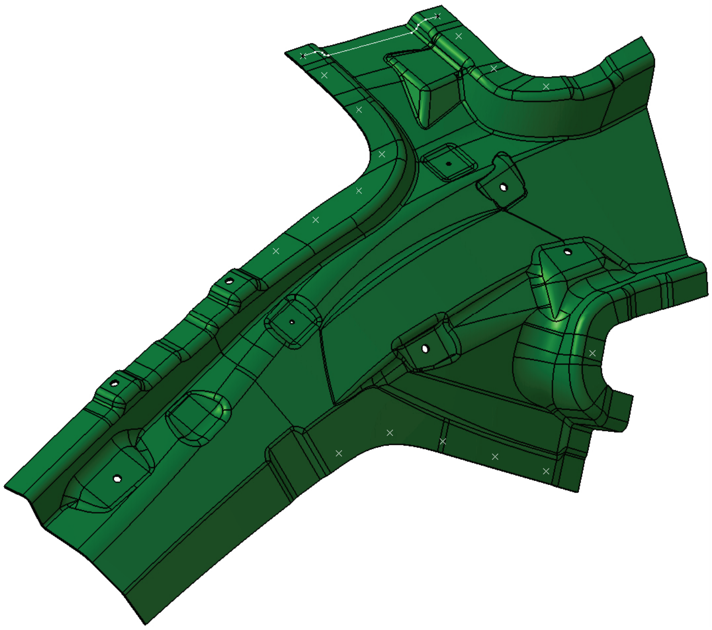


Fig. 6.23 Completed solid model inner panel rear joint sidewall upper with connecting elements (acc. to Bode et al, 2010)

6.4 Conclusion

At the university the emulation of the cooperation's between development partners such as OEMs and engineering suppliers is restricted because of the lack of PDM system. Nevertheless most of the questions regarding collaborative parametric associative design of assemblies of the automotive body can be investigated and optimised. In industry the file based approach used at the university is used in the concept phase. Under support of final projects (bachelor and master thesis) worked out in industry where students of the university assist the elaboration and verification of interrelated product development the author's findings were applied and verified. In this chapter the requirements of collaborative work for concept design with zone based approach have been discussed in detail. For the development of automotive body parts the use of sections is essential. Next chapter will describe and verify methodologies and principles of this subject. The addendum will deliver a selection of CAD-model examples which have been necessary to work out and verify the knowledge described so far.

Chapter 7 Representation of Profiles and Sections

7.1 Introduction to the Chapter

Parts of an automotive body can be classified according to their material, their manufacturing method, function or their geometry.

When an automotive body part is analysed according to its geometry it is noticed that parts are curved in one or two planes and have a common position in design space. Areas of body parts designed for shell construction or complete parts designed for space frame construction are of constant profiles (prismatic). Figure 7.1 shows a true most (radial) concept section of an A-pillar with three prismatic profile areas.

Profiles without constant cross section are designed with production intent in mind i.e. according to the manufacturing technology employed. As automotive body parts are designed in post assembly (carline) and not in manufacturing position specific design methods are necessary to develop prismatic and non prismatic profile areas.

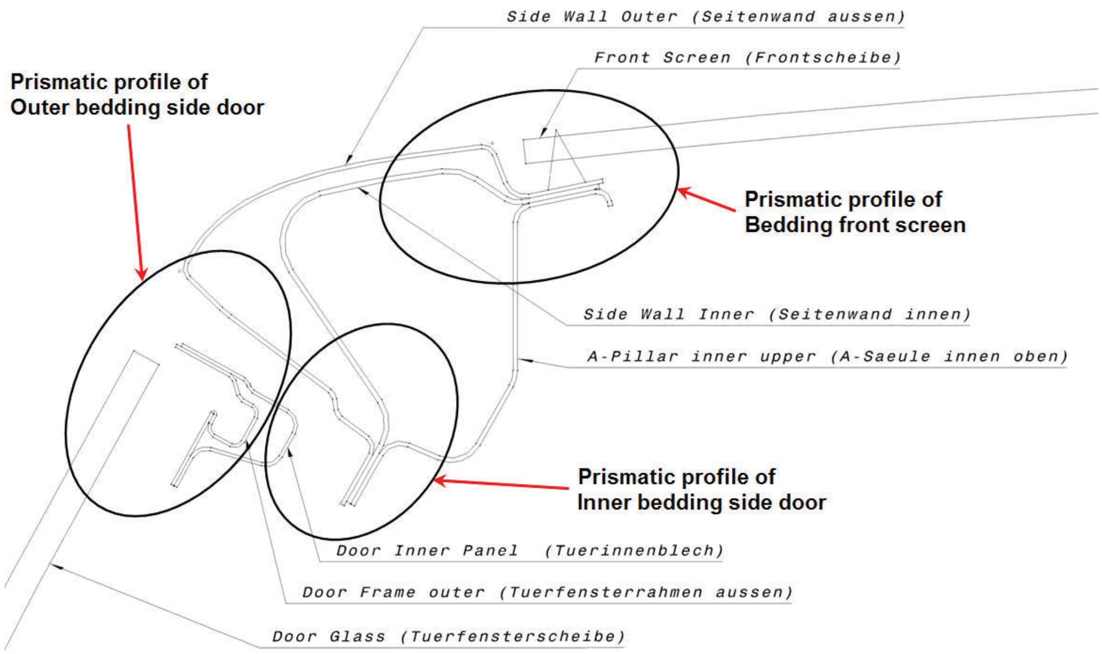


Fig. 7.1 Example of use A-pillar: True most concept section of prismatic profile areas bedding front screen, inner and outer bedding side door.

In this section most important design methods are explained and evaluated on basis of three different case studies. The three case studies (workbenches) have been defined to represent three different degrees of difficulty.

All common design methods use the same references such as a reference surface, a spine and a guide curve. For prismatic profile areas a secant surface (surface band) must be designed according to the projected width of the prismatic area to become the reference surface. In most cases a true (compound radial) section normal to reference curve is designed, to define the dependencies of the new geometry in relation to the reference surface and subsequent control surfaces. When the profiles route is expected to be prismatic, spine and guide curves must be identical. For Computer Aided Design (CAD) modelling the guide curve must be tangent and the spinal curve curvature continuous.

Manual body design applies three design methods to design prismatic profile areas. a) In the “normal plane method” the profile cutting plane is defined in 3D on the basis of two intersecting lines. Then in two different views a straight line is defined in the same intersection point normal to the reference curve. b) In the “projecting and the tangential plane methods” the profile cutting plane is defined in true view of the reference curve. To obtain the true view of the reference curve under the support of the projecting plane method, a projecting plane (normal to view plane) is defined tangential to the reference curve in its 'true most'' view. In the 'true' view of this plane the reference curve is to be seen partly true. c) In the tangential plane method a tangential plane to the reference surface is defined in any point of the reference curve. In the true view of the tangential plane the reference curve can be seen partly true. All above manual methods have in common one spine and guide curve, a normal plane and a reference surface.

In CAD design methods are established on the basis of profile surfaces or surface bands. Surface bands can either be generated on the basis of true radial compound sections (“True section”) or on the basis of the design features of the CAD-software without using a true section (“Surface band”). Circumferential constant prismatic surfaces are modelled from profile curves which when drawn include all required radii. Generally most prismatic surfaces end up in a 'joint design' as they are designed zone the basisd. Therefore the prismatic profile of the true compound radial section without consideration of any radius representations is separated into segments. Every single segment is generating a curve for the development of a surface band. It is necessary to define generating curves of appropriate length to allow a precise delimitation or enable filleting between the two surface bands. This prolongation of surface bands in width also supports the definition of planar control curves for the modelling of surface blends in the 'joint' area. The necessary prolongation of the profile segments can be delimited under support of boundary boxes (rectangles) or boundary circles which can be defined on the section plane of the profile.

In this chapter the design process from design of early package and concept sections up to the method of true (compound radial) sections which are used to derive surface bands and parts is discussed. The analysis of the several principles and practices used in CAD for the design of prismatic profile areas and the profile areas designed according to functional demands is presented. Reproducing such surfaces and curves for reliable and safe updates by other users of well established CAD models is also investigated and analysed.

7.2 Design Process and Application of Profile Surfaces

During the concept development stage of automotive body assemblies there are still several occasions where breakage of the links in the associative process chain can occur. One of these breakages is generated by the use of isolated package sections. Thus an objective of this work is to define a design process from package section to surfaces and individual components/parts without breakages.

In the early phases of Product Evolution Process (PEP) the package process optimises the space allocation for the various parts (see chapter 5). To describe these space allocation cross-sections, previously defined section developments and competitors' data are used. These sections are often distorted in terms of their geometric integrity and therefore can not be utilised for dimensioning and surface derivation and extraction. Sections are mainly grid sections (cut on a plane parallel to the vehicle XYZ-coordinate system). Often these sections are taken over from another OEMs vehicle concept or are designed on the basis of demands of competition. The sections are often not true, and do not exactly fit the styling of the new car. Therefore they cannot be used for the development of surfaces and parts of the new car. Their only purpose is to give the concept engineers an idea what kind of parts and geometries are involved in the various package spaces. OEMs often use the same design principle in packaging components for several models and variants of the brand. Profiles will only have to be adjusted from car to car according to their shape and dimensions.

Sections needed for the development of surface or solid geometry components need to be true compound radial sections. This means that the section is defined on a plane situated perpendicular to a spine curve (reference geometry).

Most concept sections contain more than one true profile with different reference surfaces (e.g. A concept section of an upper A-pillar contains a bedding profile for the wind screen, a bedding profile for the front door, Class A surfaces to the outside, functional surfaces to the inside of the car etc. (see figure 7.1)). This means that a concept section has to be subdivided in several true profiles. Several true profiles result in several primary surfaces which are completed to create parts by trimming or filleting operations.

To allow the variation of references and safe handling in updating the profiles the surface generating segments have to be designed longer by a factor of 2 to 3 than shown in the concept section. The development of true sections from the concept sections is a long cumbersome process. Therefore it is necessary to develop methodologies and principles within a process chain which facilitate an update safe environment for automotive body parts. In this chapter the process chain for the design of body parts from true most concept sections to true sections is investigated. The core area of development is the design of prismatic profiles. The true most (radial) section and true (compound radial) sections are used to design a specific zone (i.e. the upper A- or C-pillar) to prove both the legal and functional requirements of the concept. With the aid of specific case studies this chapter describes the following methodologies:

- "True section" methodology developing true most radial section and true radial compound sections for the design of surface bands and body parts

- six principles for the extension of surface generating segments in true sections
- “Surface band” methodology developing true most radial section and design of surface bands and body parts without support of true radial compound sections
- methodology for designing profile surfaces for production intend parts i.e. to fulfil tooling demands
- methodology for creating parametric associative package sections

Figure 7.1 shows an example of a simplified model for an upper A-pillar with front door and front screen beds in a radial section. This parametric associative section was designed having as a basis a distorted package section (principle section). The upper A-pillar is built from Class-A, prismatic and functional surfaces of glass barrel (retractable window, door) or control surfaces of binocular obstruction. Figure 7.2 shows a C-pillar with its concept section and three prismatic profiles on each side. These are the bedding profiles on sidewall outer, the reinforcement C-pillar and the sidewall inner for the rear door and the rear window.

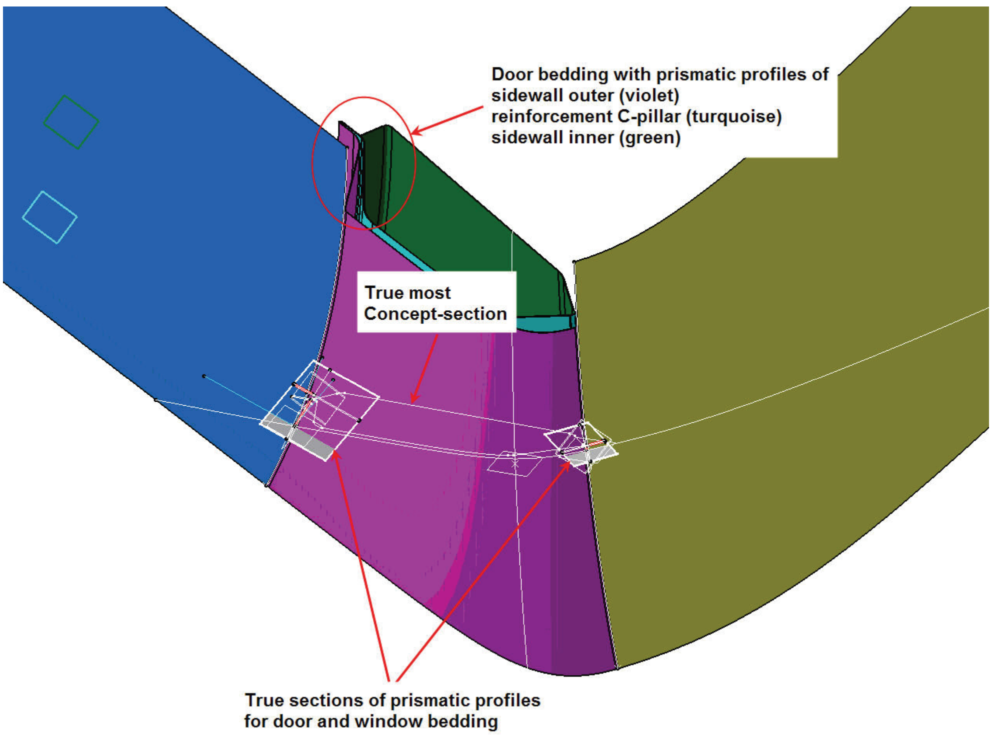


Fig. 7.2 Example workbench “C-pillar” with true most concept section and two prismatic bedding areas.

7.3 Methodology – Prismatic Surfaces “True Section”

7.3.1 Introduction

A surface area is prismatic (of constant cross section) as long as both spine and guide curve are identical and its cross section does not change along the route of the spine and guide curve. Joints are often developed after the prismatic surfaces have been designed. Therefore surface bands are derived which allow individual joint designs. For closed spine and guide curves or for the design of prismatic swages true profiles with all radii representations can be used to design prismatic surface areas.

Prismatic surface areas are differentiated according to bedding, collar and swage surfaces. Bedding surfaces (see figure 7.1) are surface areas where two or more parts define a prismatic area and the parts are connected to each other e.g. window bedding: flanges with bent portions of sidewall outer, inner and A-pillar inner define the BIW flange. Glass, adhesive, trim strip, gaps and flange profiles of BIW define the bedding; inner door bedding: flanges with bent portions of sidewall outer, inner and A-pillar inner define the BIW flange. BIW flange, weather strip, door inner panel and gaps define the bedding. Collar surfaces are mostly one segment bent portions of surfaces designed to reinforce the rim of a part. Swage surfaces (see figure 7.3) are used to avoid drumming noises of sheet metal construction, for the local reinforcement of sheet metal parts (deep draw technology) and sometimes for styling purposes

Position and shape of bedding profiles depend on their individual installation. Class A surfaces and curves, glass barrel, functions of mating parts etc., collar and swage profiles depend on the manufacturability of a single part. The manufacturing method and the necessary demould angle are have to be taken into account (e.g. sheet metal construction St requires 3°, Al requires 7°). Collar and swage surfaces differ if surfaces are designed according to manufacturing principles or are modelled as prismatic surfaces. Collar and swage surfaces can only be prismatic as long as their profile keeps constant, has a constant angle to the reference surface and their spine and guide curves are identical.

Bedding profiles defined by two or more parts, with two ore more segments for each part are designed by their bedding functions. Besides the bedding functions bedding profiles must be manufacturable. Therefore the bedding function determines the appropriate tooling angle in such a way to achieve tool release and avoid expensive tool slides.

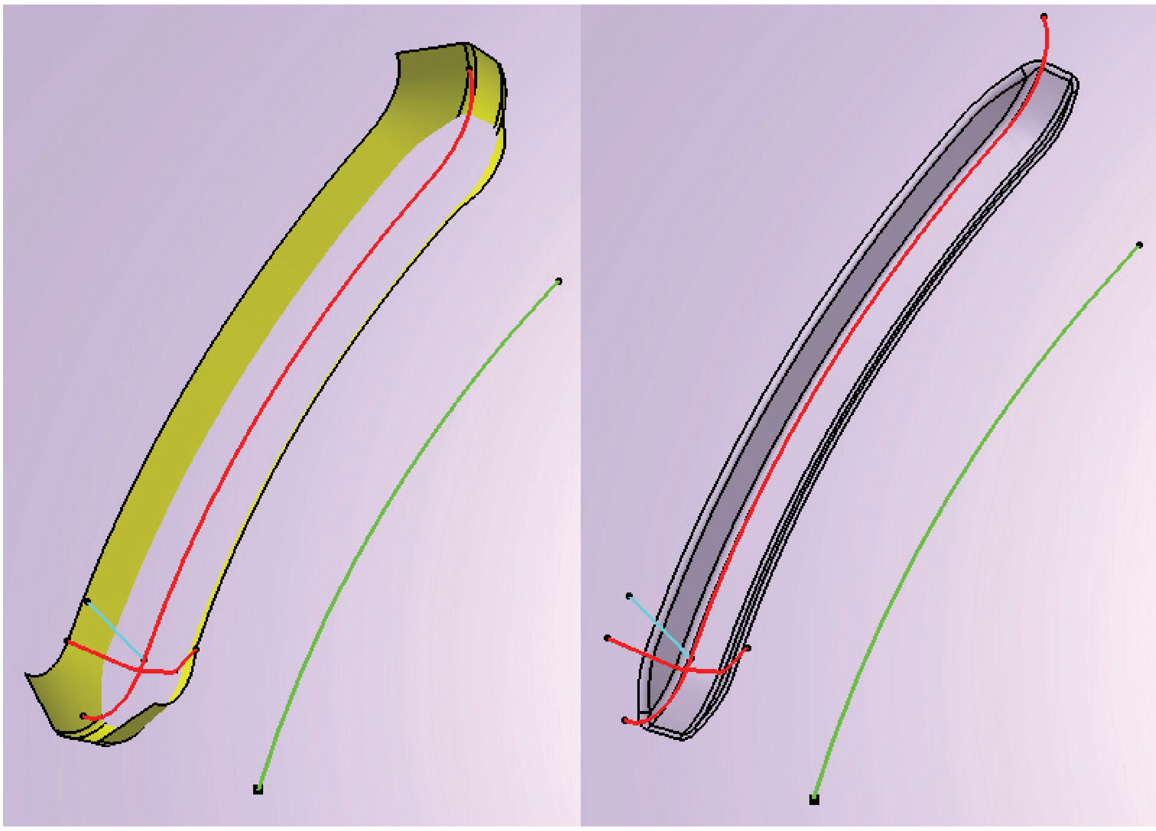


Fig. 7.3 Example of use trapezium swage with rounded run out according to Volkswagen standard 01080

Very often prismatic designs of trapezium or round swages are used for stiffening purposes. While the trapezium cross section with its slope angles at 45° or 60° is easy releasable, the circular cross section of the round swage with its small end tangent angles on the border between swage and reference surface may cause local release problems (see figure 7.4).

All automotive body part surfaces which are not directly involved in bedding functions, must fulfil the technical specifications of tooling without slides. Therefore they should not be prismatic. Surfaces designed to facilitate tool release are modelled by a spine curve on the tooling plane and the guide curve on the reference surface. These surfaces do not have a constant cross section and are not prismatic.

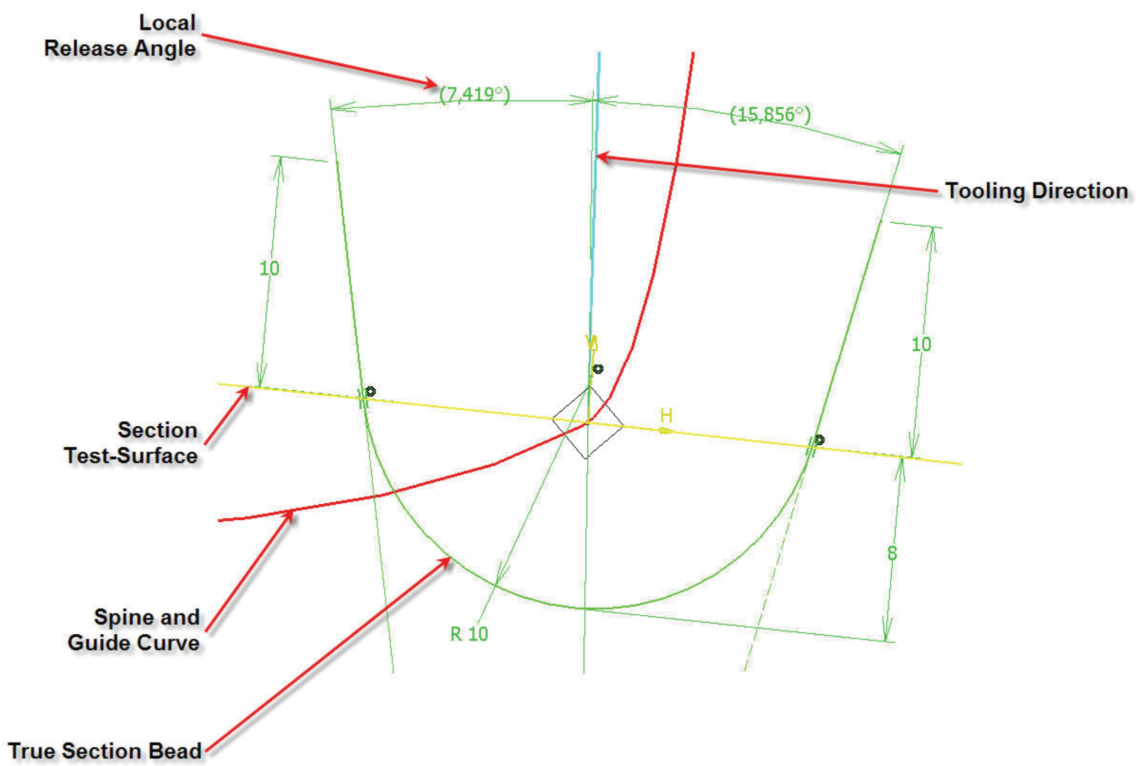


Fig. 7.4 True section of a prismatic round swage with corresponding local release angles.

For prismatic surface design according to the "true section" methodology firstly a true (compound radial) section is developed (see figure 7.4). The modelling of this section in *CATIA* is either carried out in 3D modelling space *(Work-on-Support)* or in the workbench *Sketcher*. Every surface generating segment of the section must be extended as much as possible to ensure delimitation or filleting of its generated surface with adjoining surfaces.

The prismatic bedding profile normally consists of two to three segments. Generally these segments are straight lines and the surfaces generated from these segments are skewed surfaces. Sometimes special tooling or styling requirements produce curved segments which generate arched surfaces.

7.3.2 Design of true (compound radial) Section

To model a true section first of all a normal plane is defined in any point of the spine and guide curve of the prismatic area. A true most concept section point for the normal plane could be the intersection point of the true most section plane and the spine and guide curve of the true section. If there is no true most section, the position of the true section should be chosen in a way that the whole width of the true section intersects with the reference surface. In *CATIA* the true section can be designed in 3D modelling space (e.g. with *Work on Support*) or within the workbench *Sketcher*. In 3D modelling space the true section is defined under support of a local axis system in the workbench *Generative Shape Design (GSD)*. In workbench *Sketcher* the true section is positioned according to the local axis system (see figure 7.5).

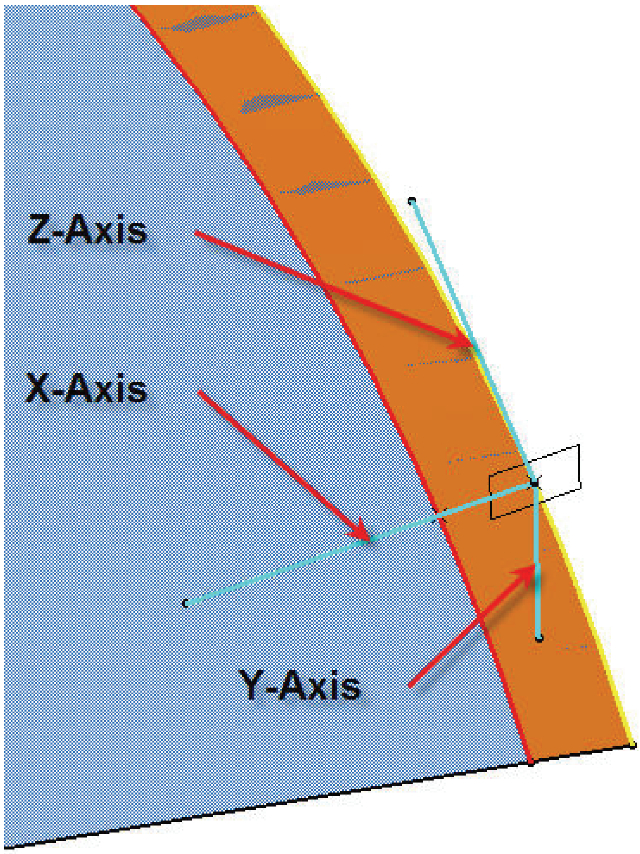


Fig. 7.5 The local axis system is the basis for the true section

7.3.2.1 Secant Surface (Reference Surface)

The projecting width of the prismatic profile (true section) defines the distance between the two parallel curves, spine and guide curve and a parallel curve, defined with the equidistant distance on the Class-A surface provided as input of the design. At the end of both parallel curves, a straight line (secant) connects both curves. The straight line and the two parallel curves generate the secant surface (reference surface) for the design of a prismatic surface.

7.3.2.2 Local Axis System

All methodologies utilise a local axis system (see figure 7.5) for the positioning of the true section. For the definition of the X-axis the section plane is intersected twice with the spine and guide curve as well as with the parallel curve of the secant surface (reference surface). The two intersection points define the X-axis (H-axis, horizontal axis of the section) which determines the width of the section. The Y-axis (height and vertical axis, V-axis, of the section) is defined perpendicular to the X-axis on the section plane. The Z-axis (generating direction of the prismatic surface area) is a tangent to the spine and guide curve.

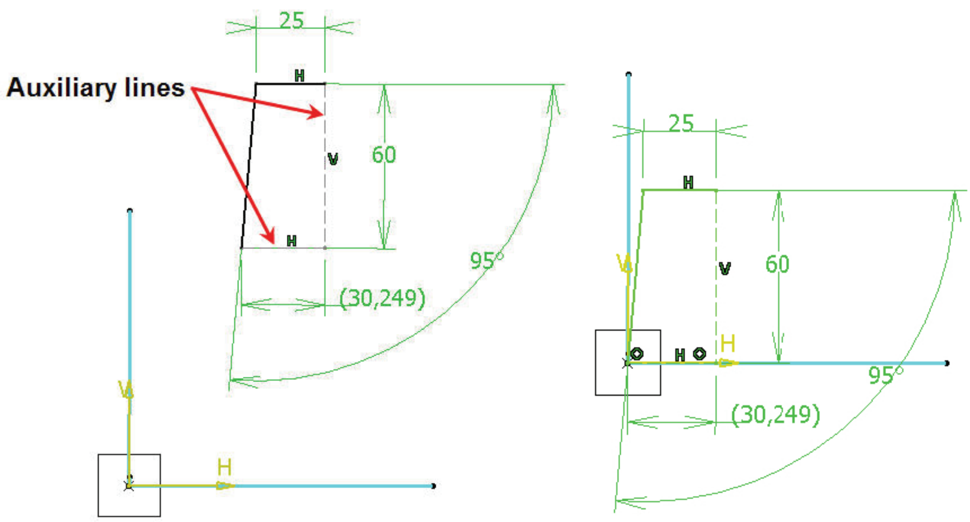


Fig. 7.6 The separate design of the outer profile is completed with auxiliary lines before it is linked to the local axis system

7.3.2.3 Positioning of the prismatic Profile

In 3D modelling space or in workbench *Sketcher* the true section is positioned utilising the local axis system (see figure 7.6). The outer bedding profile (e.g. door bedding, sidewall outer plus gap plus window frame door) defines the projecting width of the prismatic bedding. This profile first is designed without regard to the local axis system. Two auxiliary lines parallel and perpendicular to the flange segment complete the open profile to a trapezium. The prismatic L-profile generally is constant. In a second step the profile is connected to the axis system of the Sketcher (local axis system).

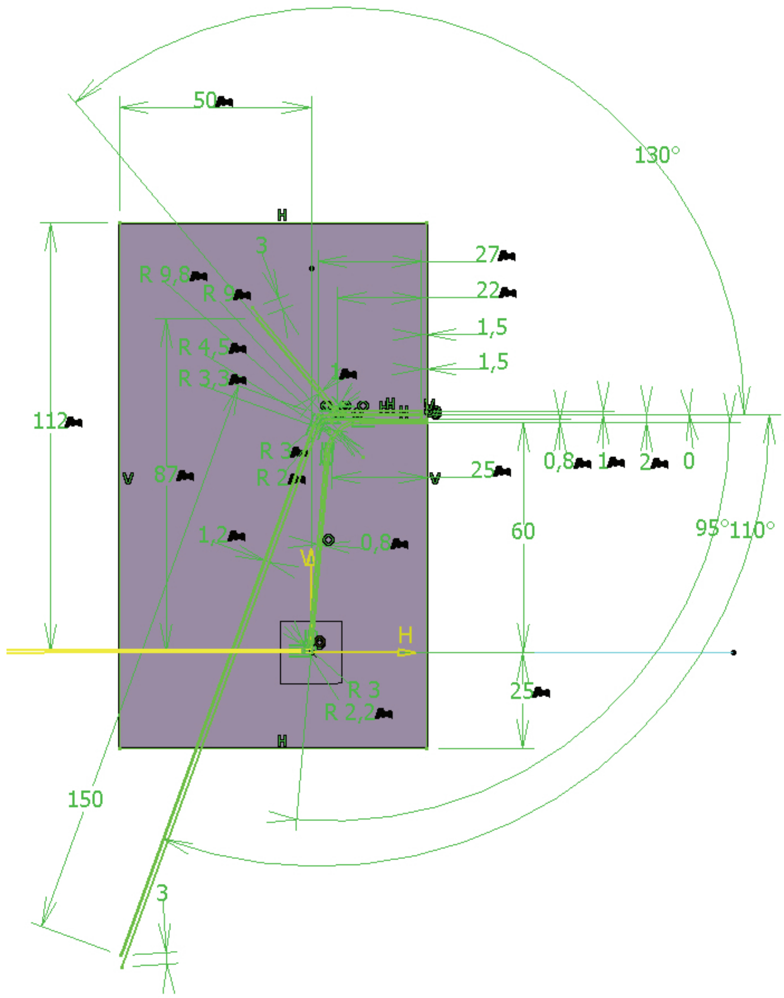


Fig. 7.7 Completely dimensioned true sections (Workbench *Sketcher*, principle "Boundary box (surface)" of door bedding, sidewall outer, reinforcement C-pillar, and sidewall inner. Bent portion, reinforcement C-pillar is extended under support of dim. 150. All other segments are not extended.

After positioning of the first prismatic profile the next profiles of bedding are designed according to the first profile (see figure 7.7- e.g. door bedding, sidewall outer, reinforcement C-pillar and sidewall inner). All dimensions are defined relative to the local axis system (fully constrained). When the true section is designed in workbench *Sketcher* the consistency of the section must be checked by *Sketch Analyse* during the design and after completion.

Important control parameters (e.g. secant length, flange length, sheet metal thickness) are explicit parameters defined in the hierarchical tree of the CAD-model and linked to the true section and the design features. All other dimensions of the true section are linked to the corresponding constrains of the true most concept section. First of all in the design of the section only one side (design side) of the sheet metal, the Class-A side for all visible body parts and the punch side for invisible body parts is designed and constrained. The sheet thickness to be shown in the true section is defined by parallel curves and radii (manager section). Only two design principles ((2) and (6) of the list stated in Section 7.2.3) allow the representation of sheet thickness in the true section. If the true section is designed in this order the design of surfaces and fillet operations will not lead to errors in updating.

7.3.3 Principles for the Extension of Surface Generating Segments

To assure update safe parametric associative design of the surfaces it is necessary that the corresponding surface bands of every part designed allow a precise delimitation or enable filleting between two surface bands even after any possible alteration of reference geometries and parameters. Therefore the surface generating segments of the true section have to be extended in 3D modelling space or in workbench *Sketcher* before surface bands are generated.

Apart from two principles where the extension is carried out in 3D modelling space or in workbench *Sketcher* by dimensioning, three more principles are investigated which work under support of auxiliary geometry. Apart from these five methods it is possible to carry out the prolongation by extrapolation in 3D modelling space. In most cases it is necessary to design auxiliary geometry for the prolongation to avoid the use of *BRep* elements. Figure 7.7 shows a prismatic profile consisting of two segments, a flange and a bent portion. A flange is only extended on the side where it is to be filleted with the bent portion. Bent portions are extended on both sides.

The following principles for the design and extension of surface generating segments were investigated:

(I) The true section as well as the extension of the segments is designed in 3D modelling space.

(II) The true section is designed in workbench *Sketcher*. The extension takes place in that workbench by constraining.

(III) The true section is designed in workbench *Sketcher*. The extension takes place in that workbench by boundary circles.

(IV) The true section is designed in workbench *Sketcher*. The extension takes place in that workbench by a boundary box (wire frame).

(V) The true section is designed in workbench *Sketcher*. The extension takes place in 3D modelling space by a boundary box (surface).

(VI) The true section is designed in workbench *Sketcher*. The extensions take place in 3D modelling space.

In addition the prismatic design of swages where investigated. Examples of swage design are described in chapter 8. For the extension of the swage segments the third of the above listed principles is used.

The detailed evaluation of the methods is presented at the end of the description of all methodologies and principles.

7.3.3.1 Design of true Section and Prolongation in 3D Modelling Space

For this principle the whole design takes place in 3D modelling space (see figure 7.8). All design features used are visible in the hierarchical tree including their numerical parameters.

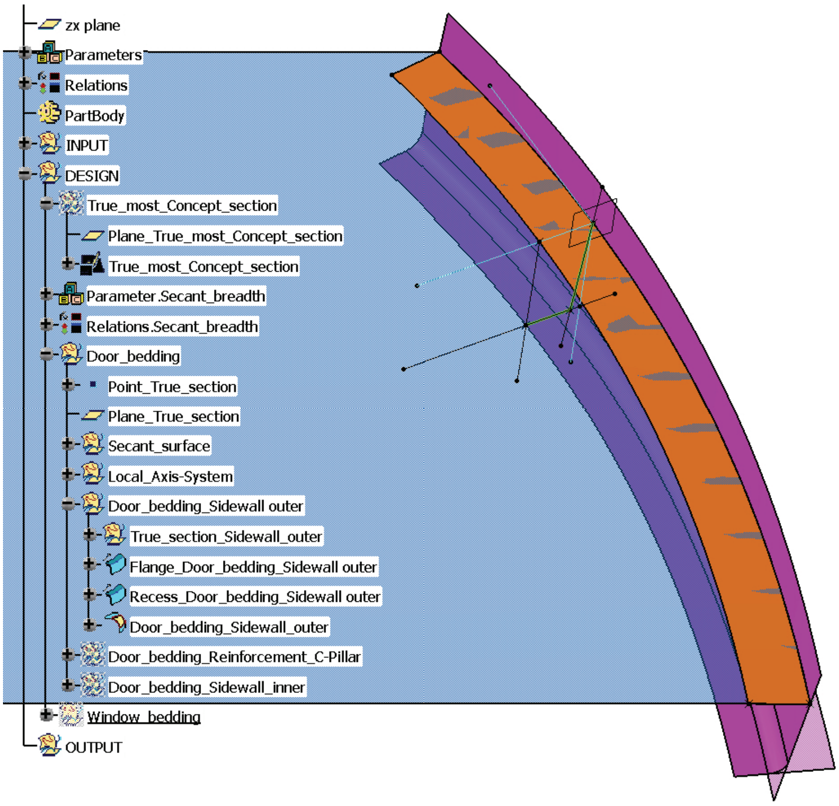


Fig. 7.8 Complete design of true section and prolongation of segments by extrapolation in 3D modelling space (example: door bedding, sidewall outer)

On basis of the local X- and Y-axes of the local axis system the single segments are designed consecutive in the form of straight lines, circles, 3D splines; offset curves etc. on plane true section. In *CATIA Work-on-Support* can be used. Length of segment and prolongation can be defined in the pull down menu of the single design feature.

A design of the segments in their original length and a prolongation by extrapolation is possible. To avoid the use of BReps points are defined on the ends of the segments where an extension is necessary. This principle is possible for straight as well as for curved segments.

The test to replicate the principle from bedding profile to bedding profile was not successful. The effort in time to adjust the replicated design features is as high as the redesign process as requirements may be different from profile to profile.

7.3.3.2 Design of True Section in Workbench Sketcher – Prolongation by Constraining in Workbench Sketcher

Firstly all the bedding profiles are designed in their original lengths. At the end of every segment needing extension a point is defined. A geometrical constraint (coincidence) is defined to position the point onto the direction of the segment. A dimension between the new point and the end point of the segment is placed to describe the required extension. A trim-operation for the segment results in the expected prolongation (see figure 7.9).

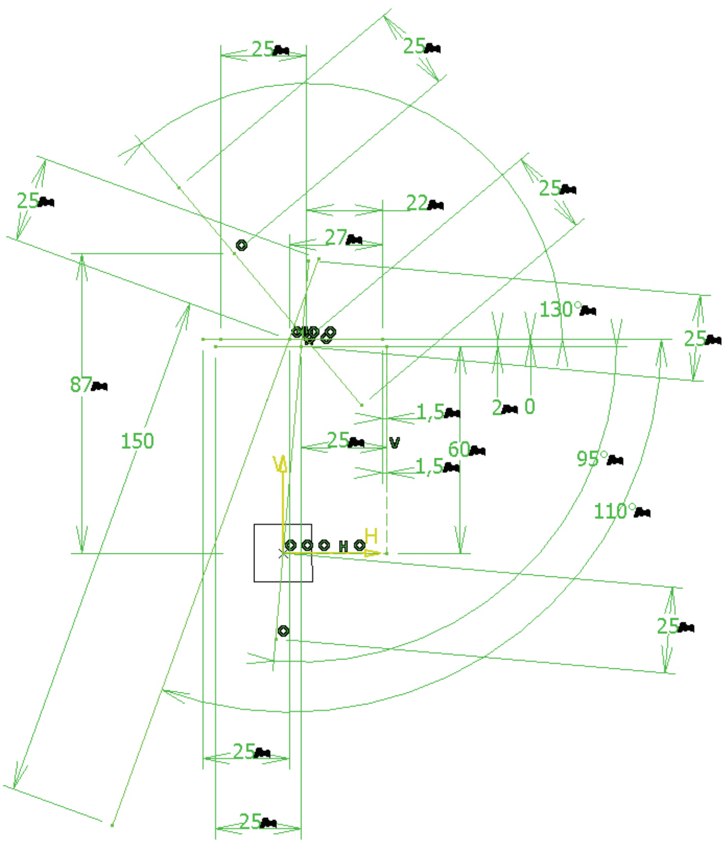


Fig. 7.9 Prolongation of segments under support of auxiliary points defined with distance to the endpoints of segments

Curved segments are designed by three points as an arc. For prolongation parallel auxiliary lines to the local axis system are defined in the required distance to the original end point of the segment. A trim-operation for the segment results in the expected prolongation.

A test to replicate the complete principle of profile design, prolongation and design of surface bands carried out for the door bedding in order to generate the window bedding was update safe and repeatable.

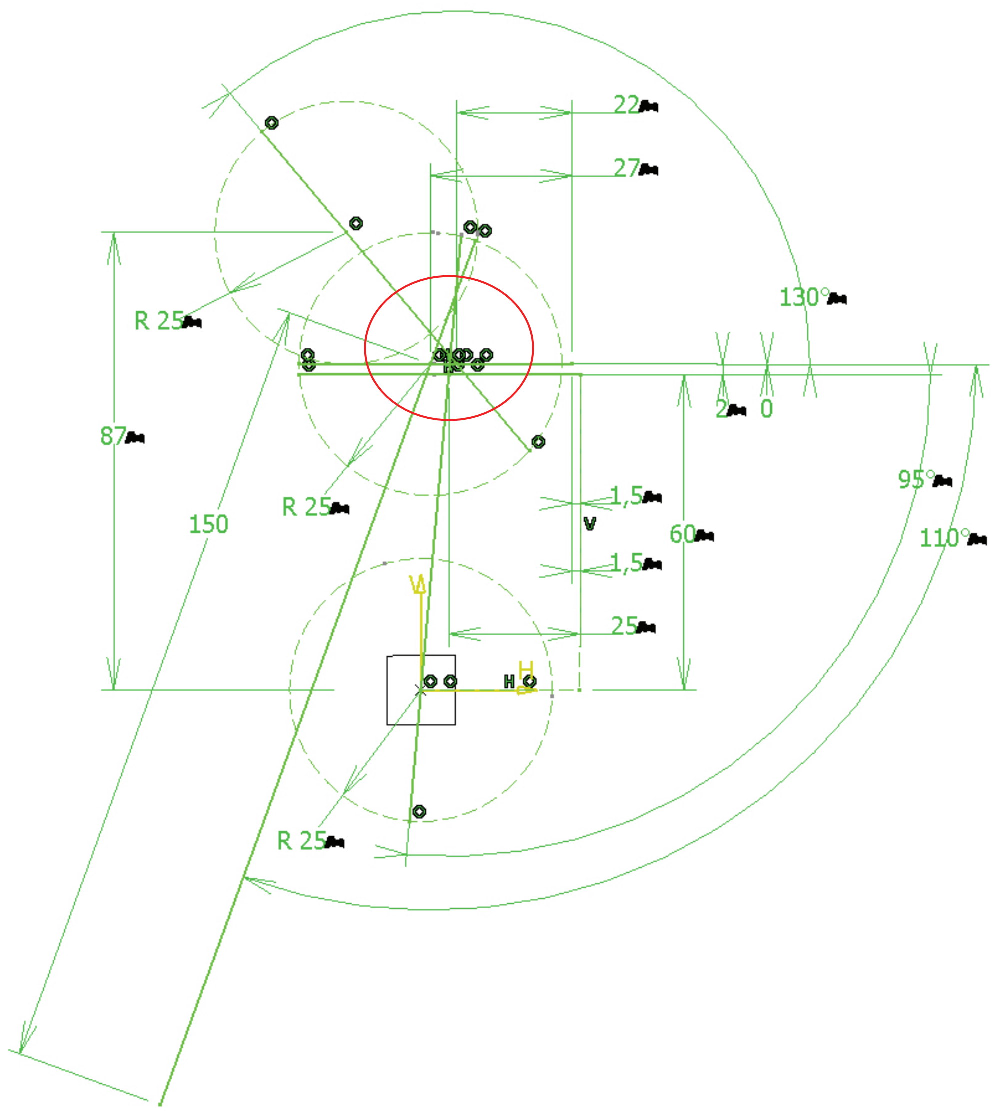


Fig. 7.10 Using of boundary-circles defined in intersection or end points of segments, the profile segments are extended. In the centre only one of three intersection points was chosen to avoid complexity.

7.3.3.3 Design of true Section in Workbench Sketcher – Prolongation by "Boundary Circles" in Workbench Sketcher

At every end of a segment required to be extended a circle is defined. The radius of the circle satisfies the expected prolongation length. As a flange and a bent portion intersect each other only one circle is designed in the intersection point in order to extend two segments. Flanges are not extended at the flange end. Bent portions are extended on both sides. A trim-operation between segment and auxiliary geometry results in the expected prolongation.

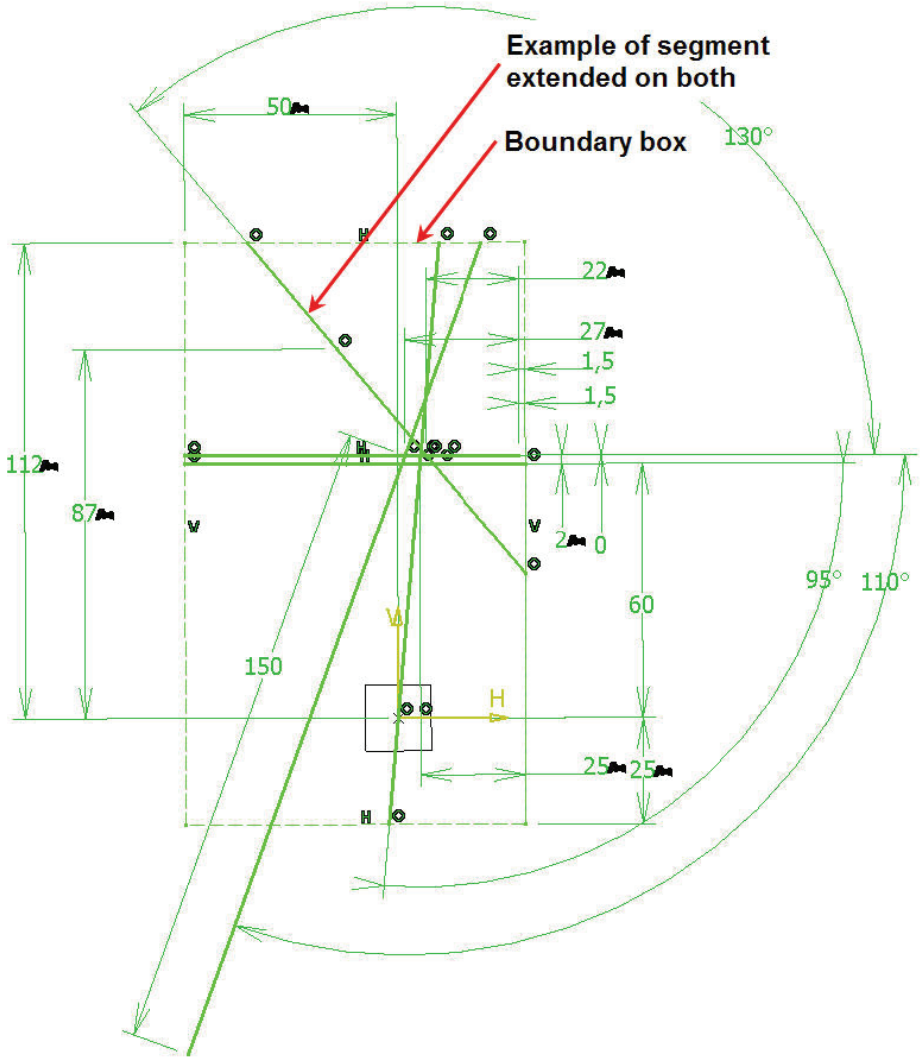


Fig. 7.11 Using of a rectangular boundary-box defined relative to flange end and local axis system the segments are extended.

Theoretically in the example of figure 7.10 a door bedding with three pairs of flange and bent portion six auxiliary circles are necessary to prolong the segments. As the intersection points of flanges and bent portions are very close to each other only one intersection point is chosen to avoid unnecessary complexity.

This principle can be used for straight as well as for curved segments. For extensive profiles many auxiliary circles have to be designed which results in unnecessary complexity. The geometrical constraints defined by the trim-operations results in another form of complexity when segments are very close to each other. Updates tests have shown that the geometrical constraints defined with the trim operations are not always update safe. For this reason this principle is only recommended for simple profile situations.

7.3.3.4 Design of true Section in Workbench Sketcher – Prolongation by "Boundary Box (Wire Frame)" in Workbench Sketcher

In this principle auxiliary lines define a rectangle in the true section (see figure 7.11). The lines are defined at the end of flange of the outer profile and parallel to the local axis system. The auxiliary geometry can also support the design and the dimensioning of the profiles involved. Therefore it is necessary to design the rectangle directly after design and positioning of the first profile (see figure 7.6). Trim-operations between segments and auxiliary geometry lead to the expected prolongations. This principle can be applied to straight and curved segments. In the example shown in figure 7.11 the principle is clearly arranged and reproducible. Update tests have shown that geometrical constraints defined with the trim operations are not always update safe.

7.3.3.5 Design of true Section in Workbench Sketcher – Prolongation by "Boundary Box (Surface)" in 3D Modelling Space

Similar to the principle above the true section is defined according to its original parameters. A rectangle defined by auxiliary lines and positioned relative to the flange and local axis system is the foundation of this prolongation approach. In 3D modelling space a bounded surface (face) is designed by a trim operation between plane true section and rectangle. Planes are positioned perpendicular to the plane true section on every straight segment. The intersections between the bounded surface and the normal planes are the extended segments (see figure 7.12).

This principle allows extending straight segments without the use of unstable geometrical constraints. Curved segments must be extended according to principle “boundary-box (wire frame)”. As all segments will be extended by this principle flanges must be shortened again with additional effort. The principle provides dressing up the true section with sheet thickness illustration (manager section) by parallel curves and radii.

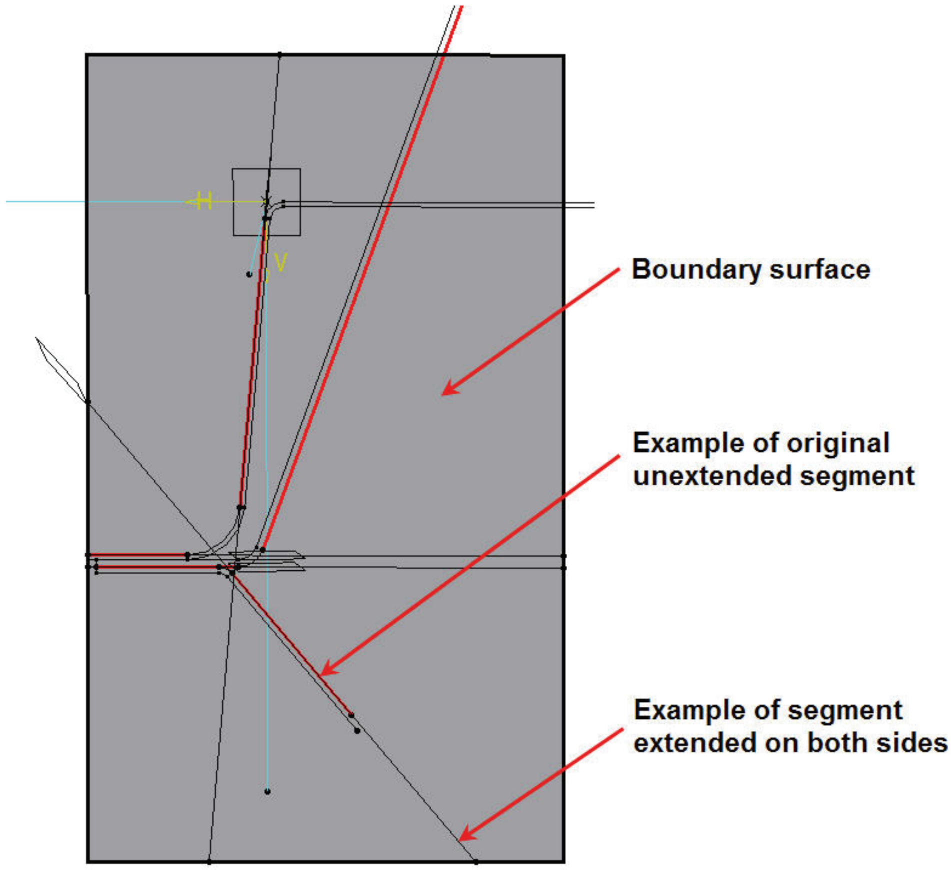


Fig. 7.12 Using a boundary surface segments are extended in 3D modelling space.

At first glance this principle seems to be very complex. As several design operations reoccur, a lot of design effort can be reduced by replication of defined operations. For easy replication design operations must be composed in usable files *(Geometrical sets).*

7.3.3.6 Design of true Section in Workbench Sketcher – Prolongation in 3D Modelling Space

The design of the bedding profiles in their original lengths takes place in workbench *Sketcher* (2D) Segments and endpoints defined in the true section are used to extrapolate the segments in 3D modelling space. This approach needs clearly arranged denominations and attention not to mix up points and segments for the extrapolations.
The principle allows dressing up the true section with sheet thickness illustration (manager section) by parallel curves and radii (as in 7.2.3.5).

The design of the true section and the prolongation can not be reproduced as the single design steps are inherently different.

7.3.4 Design of Surface Bands and Completion of Design Zone

From every extended segment curve, a surface band is generated under support of identical spine and guide curve and reference surface (in most cases the secant surface). While in manual design a number of true sections according to the length and curvature of the spine and guide curve are necessary to display prismatic surfaces in the form of sections and connecting curves (wire frame geometry), in CAD design only one true section is necessary to sweep the entire surface band.

For a typical bedding situation the design of surface bands has to be repeated with all segment curves. A lot of effort can be reduced when the first swept surface is replicated by replacement of the generating segment curve. To prepare the surface bands for follow-on operations such as filleting, the direction of the surfaces is controlled and inverted. In the example of an upper C-pillar (see figure 7.13) the door bedding consists of three body parts involved in the bedding. The same parts need prismatic window bedding profiles on the rear side of the upper C-pillar as well as the complete design of door bedding is replicated for the design of the window bedding in order to reduce the effort of a complete new design. For the replication it is only necessary to alter parameters, denominations and some surface directions and to renew the links defined between design features and explicit parameters or the dimensions of the true most concept section.

The bedding profiles of the sidewall outer of the example in figure 7.13 are filleted with the Class-A surface (purple). For the reinforcement C-pillar an offset of the Class-A surface is filleted with its bedding profiles (blue-grey). On the bent portions of the inner sidewall profiles parallel curves to the flanges are the basis for the development of the closing surface between door and window bedding. A skewed surface between these curves is designed and filleted to complete the zone (green).

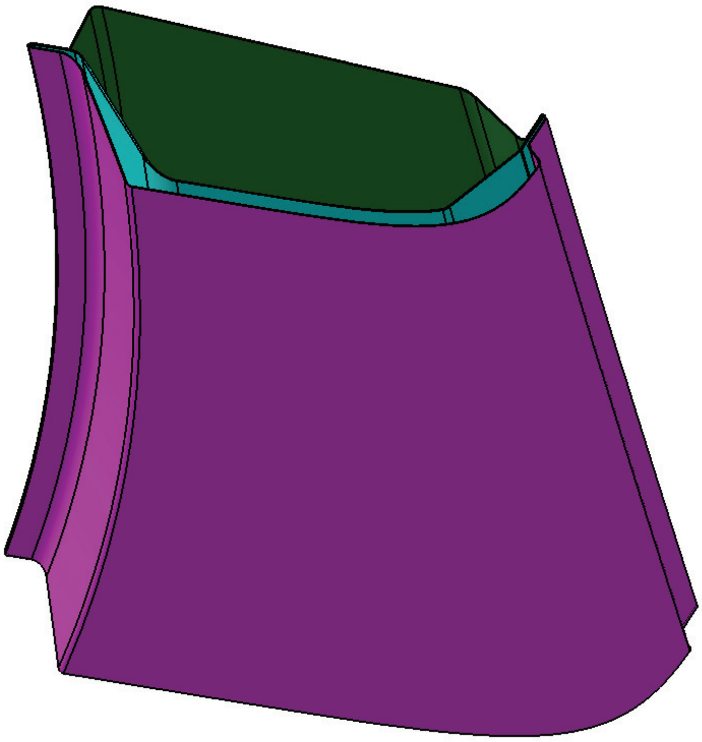


Fig. 7.13 Example zone C-pillar, upper with door bedding on the left and window bedding on the right

7.4 Methodology – Prismatic Surfaces “Surface Band”

7.4.1 Introduction

With this methodology the prismatic profile surfaces are designed without the previous design of a true (compound radial) section. This methodology can be often found in the design works of automotive body designers. Starting from a reference surface (secant surface) all surface bands are designed one after another using the last surface band as reference (see figure 7.14). The secant surface is based on a parallel curve (Curve 1) on the Class A surface. The flange is generated by an offset from the secant surface and bounded by two parallel curves (Curve 2 and 3) to the projection of Curve 1. The recessed surface (slope) is designed as Sweep under support of Curve 3 and the Spine. The position of the successive surface band depends on explicit parameters, distances and angles defined in the hierarchical tree, in the true most concept section or in a simple sketch on a piece of paper. The parameters used can be reproduced on the basis of the result of the single design features used which are displayed in the hierarchical tree. Updates after alteration of parameters may lead to erroneous results because of the different directions of curves and reference surfaces. CAD-models designed according to this methodology are not easy to use as parameters and generating segment curves are not presented in a true section but kept hidden in the design features and their pull down menus. In this methodology the true (radial compound) section is the last design step – the result of the design efforts.

7.4.2 Design of Surface Bands

On the basis of the reference surface (secant surface) and the spine and guide curve surface bands are designed one after another as offset or inclined swept surfaces. When one surface band is designed it becomes the reference surface for the next surface band. Intersection curves of completed surface bands or parallel curves on surface bands deliver the spine and guide curve for the next surface band. Surface bands are trimmed by parallel curves. The pure 3D design work may lead to user errors by selection of BRep-elements instead of designed curves and surfaces. The control of parameters such as flange lengths or angles of inclination is performed in a mixture of explicit parameters, dimensions of the true most section and the values defined in the pull down menus of the design features. The width of the secant surface which controls the position of all surface bands of prismatic bedding is defined in the true most concept section.

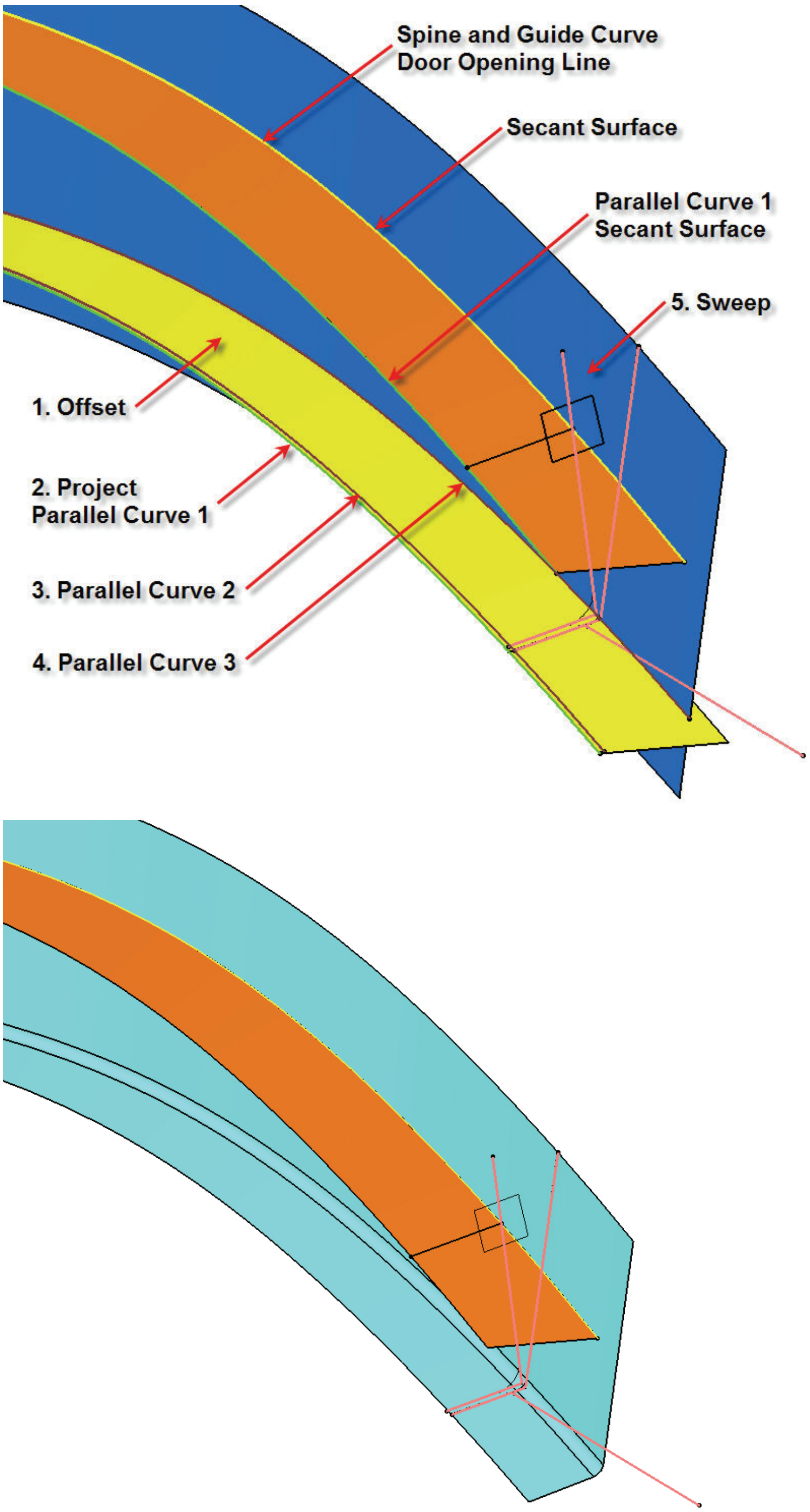


Fig. 7.14 Step by step designed surface bands of a door bedding C-pillar, upper

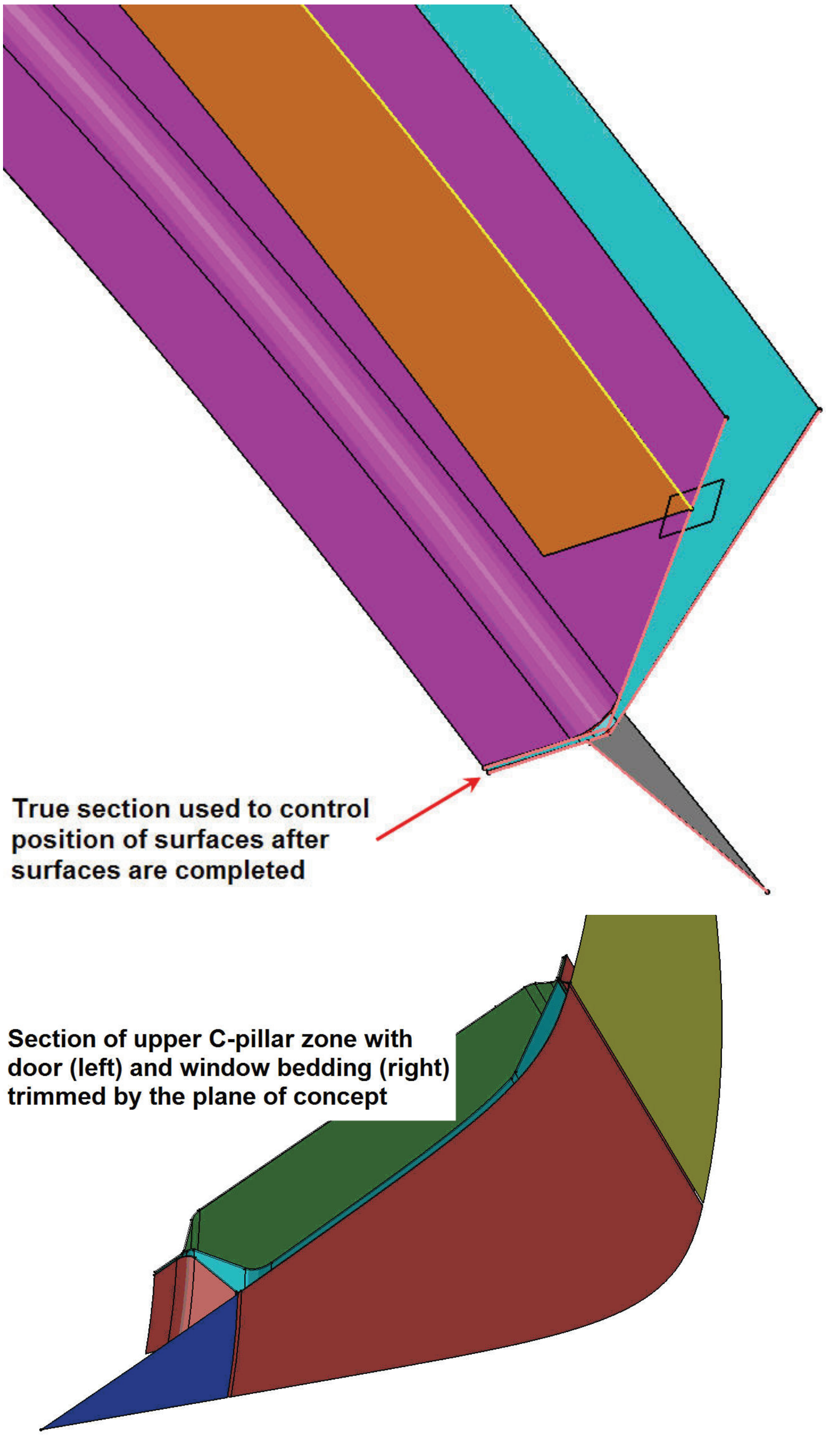


Fig. 7.15 Controlling of end result by true section compared to the true most concept section (upper C-pillar)

The length of every surface generating segment curve must be defined in the pull down menu of the single design feature, to make sure that any alteration of parameters or geometry in the designed bedding is always allowing automatic filleting or delimitation. In the pull down menus these lengths are defined according to points of the chosen spine and guide curve. A similar situation occurs with the definition of angles of inclination relative to directions of previous surface bands and a point on the spine and guide curve. The directions of curves and surfaces must be controlled during first design and after any subsequent alteration. Otherwise this methodology easily results in unwanted update results.

To control position and width of the designed surface bands compared to the true most concept section a true section is cut when all surface bands and prismatic bedding profiles are completed. This is very useful to avoid unintended deviations between the surface band and the concept section (see figure 7.15).

The update stable design of a CAD-model according to this methodology takes a lot of time because of the necessity to check and invert every surface band and curve direction involved to generate or trim surface bands. All design steps and their parameters are visible in the hierarchical tree, and apart from the use of dimensions of the true most section it is not necessary to switch from one workbench to another. Profile alterations by third parties (designers) are not easily undertaken because the dependent design steps used to develop the design model the first time can not be easily identified in the hierarchical tree.

7.4.3 Workbench Example

A prismatic design of the door bedding of an upper C-pillar is explained (see figure 7.16). Utilising the door opening line and the secant surface the prismatic sub zones of sidewall outer, reinforcement C-pillar and sidewall inner are designed.

As the design for the C-pillar and sidewall inner repute similar reinforcement the design of the reinforcement is replicated for sidewall inner under adjustment of parameters, denominations and links. The same approach of replication is used to design the complete window bedding of the upper C-pillar after the door bedding is completed.

Firstly for the design of the door bedding the secant surface (reference surface) is designed. Therefore similar to the principle used in Section 7.2.2 a parallel curve to the door opening line and a connecting straight line of the curves is defined to design the skew reference surface (see figure 7.14) .

Utilising a constant angle to the secant surface the recess surface of the door bedding, sidewall outer is designed. Based on the door opening line, the length of the generating segment is defined in the sweep-feature using dimensions of the true most concept section plus the parameter of extension. An offset of the secant surface defines the flange surface of the door bedding.

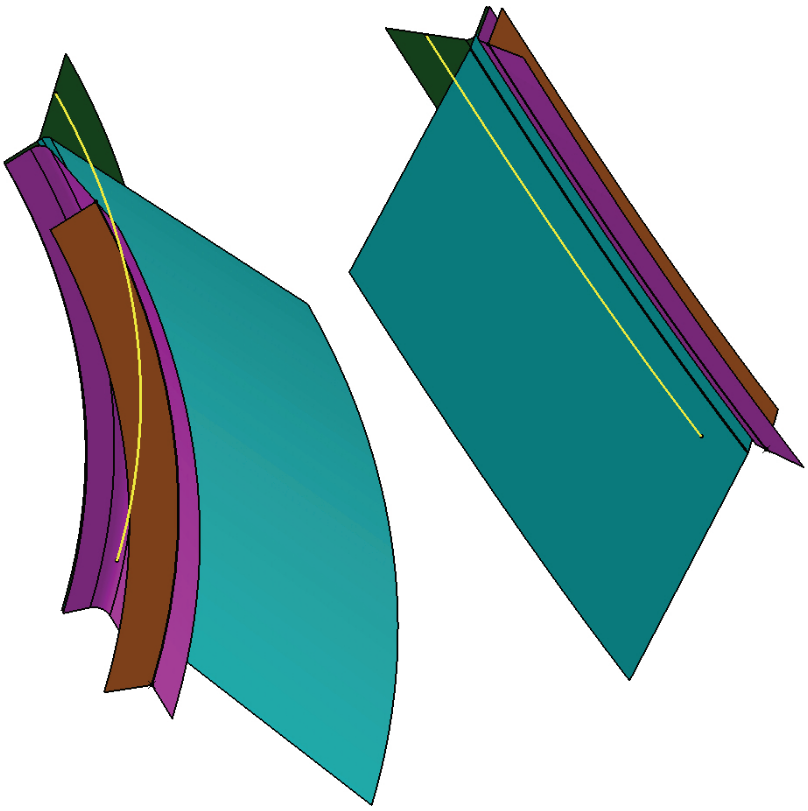


Fig. 7.16   Prismatic profiles of door bedding and window bedding

For the door bedding, reinforcement C-pillar firstly the flange surface is defined as an offset of the secant surface. The parallel curve of the secant surface is projected onto the offset. To shorten the flange according to the dimensions of the concept section another parallel curve is designed on the offset. A second parallel curve to the first with distance “flange width” (explicit parameter) is the new spine and guide curve for the bent portion. Under support of an angle constraint to the flange surface linked to the concept section the bent portion is designed.

As the door bedding, sidewall inner is going to be a similar design the modelling steps for the reinforcement C-pillar is replicated. After the alteration of parameters and colours, denominations and regeneration of links to the concept section two more prismatic surface bands are completed. Once the six surface bands of the three parts are designed, flanges and bent portions are filleted. A true section is cut through the generated profiles to compare the results in (position and dimensions) with the inputs of the concept section.

The profiles of the door bedding are replicated to design the window bedding. Once the alteration of parameters, directions, denominations and regeneration of links to the concept section the design of the window bedding is completed with minimal effort (see figure 7.16).

7.5 Methodology – Profile Surfaces for Production Intent (Tooling Requirements)

7.5.1 Introduction

In the early phase of concept development the breakdown of an automotive body into parts, sub-assemblies and assemblies is not completed. Often it is not possible to decide in this stage which parts belong to which zone. Therefore common definitions such as sidewall outer, reinforcement sidewall or sidewall inner are used instead of part denominations like side panel outer, side panel inner front or A-pillar inner upper. Similar to the denominations the dimensions of a new part may not be fully defined. As the tooling direction is defined by the dimensions and shape of a part the surfaces of the prospective part cannot always be designed according to tooling requirements.

As long as constant profile zones (prismatic surface areas) are designed imaginary part dimensions and tooling directions are sufficient. Using parametric associative methodologies the design can easily be modified to new conditions. But as soon as the design of joint areas is started the breakdown into single parts and the definition of tooling directions is necessary for the configuration of the joints.

If inside a prismatic surface area an adjustment of an angle of inclination for a bent portion is required the angle can easily be changed e.g. in the menu of the design feature or be replaced by a law which abolishes the prismatic design but allows the gradual change of the angle from one value to another. Contrary to the pure design of prismatic profiles this methodology allows the design of surfaces which may vary in width and/or angle of inclination utilising individual explicit or implicit laws in combination with *Sweep* and *Parallel-Curve* features. Sometimes this is necessary when prismatic profiles in parts are not demouldable or must be adjusted to fit a joint design.

7.5.2 Part Definition and Tooling Direction

The breakdown of an automotive body into its single parts depends on several factors such as availability of manufacturing methods, capacity of deep drawing presses, manufacturability, order of assembly, accessibility of connecting flanges for the joining technology etc.

The C-pillar of a 3 series BMW has three parts involved in the assembly of the pillar outer side panel, reinforcement C-pillar and inner side panel, C-pillar upper, have different sizes and positions in 3D space.

The outer side panel involves the A-pillar on its front end and reaches as far back as the rear lamps. At its upper end the outer side panel is connected with the roof panel and at its lower end it defines the sill and is connected to the floor panel. The tooling direction is the Y-Axis of the car axis system and the tooling plane is equivalent with the XZ-plane. The outer side panel represents the Class-A surfaces of the design. The size and shape of the outer side panel and the contradiction of terms for tooling and functions such as prismatic beddings for rear window or roof panel often lead to undercuts which have to be formed under using slide operations. The outer surface of the outer side panel is a Class-A surface. Sheet metal thickness is built from outside to inside.

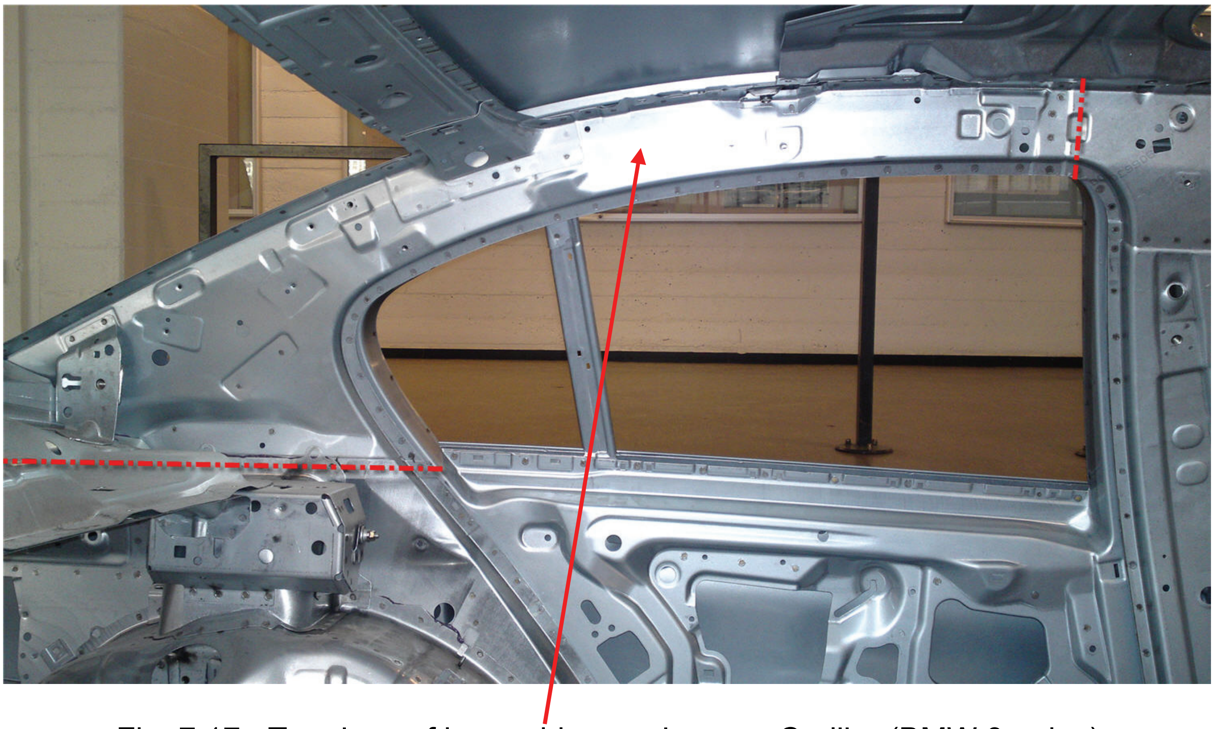


Fig. 7.17 Topology of inner side panel, upper C-pillar (BMW 3series)

The reinforcement C-pillar meets at its front end the upper end of B-pillar and at its rear end it is part of the rear window bedding. At its upper end the reinforcement is part of the lateral roof rail and at its lower end a circumferential bent flange is welded to the wheel arch. Topology and size of the part define the tooling direction and plane. The tooling plane is necessary to design surface bands with constant or variable release angle. The punch surface of the reinforcement C-pillar is designed. The topology of the part defines the punch side on the inside of the part, to which the sheet metal thickness is built from inside to outside.

The inner side panel, upper C-pillar meets at its front end the upper end of B-pillar and at its rear end it is part of the rear window bedding. At its upper end the reinforcement is part of the lateral roof rail and at its lower end the part is welded to the upper flange of the wheel arch. Topology and size of the part define the tooling direction and plane. The punch surface of the inner side panel is designed. The topology of the part defines the punch side on the outside of the part, to which the sheet metal thickness is built from outside to inside (see figure 7.17).

In the early stage of concept design only the prismatic surface areas are completed when the breakdown of the body into single parts and the definition of their tooling directions are identified. As the definition of the tooling direction depends on the shape and size of a part the definition must be carried out under support of the prismatic surface areas and placeholder geometry.

The definition of the tooling direction for symmetrical parts such as roof, front lid, tail gate etc. is not very complex. On the symmetry plane the extreme points regarding the size of the part are connected with a secant. Its perpendicular bisector of the side is the tooling direction of the symmetrical part. A normal plane to the tooling direction is the tooling plane.

On the example of the inner side panel, upper C-pillar (see figure 7.17) it is shown how the shape and the common position of the part is taken into account to define tooling direction and plane. Firstly the extreme points of the imaginary part regarding its size are connected by a first secant on the basis of the Class-A surface. In the centre point of this secant a first normal plane is defined (see figure 7.18).

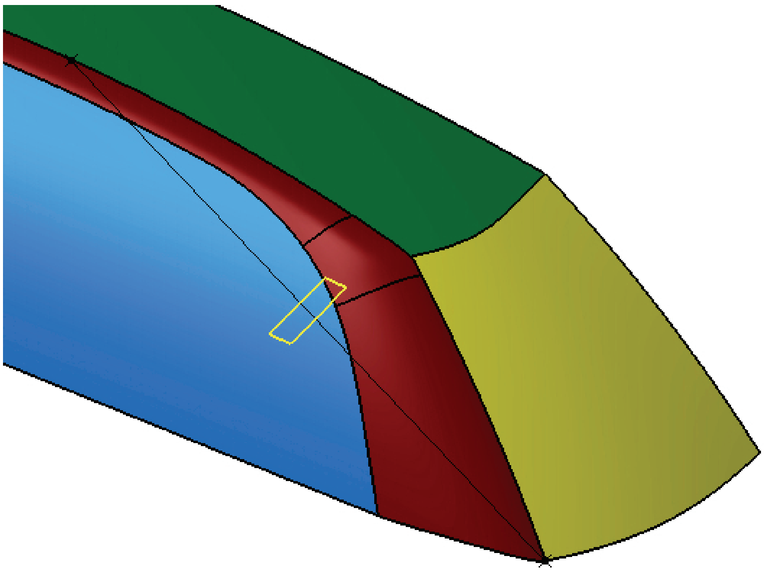


Fig. 7.18 Extreme points of the size of the imaginary side panel inner are connected by a secant and added with the first normal plane in its centre point.

The first normal plane is intersected with the concept surfaces (Class-A). At the end of the intersection curve points are defined and connected by a second secant. In the centre point of the second secant a second normal plane is defined (see figure 7.19).

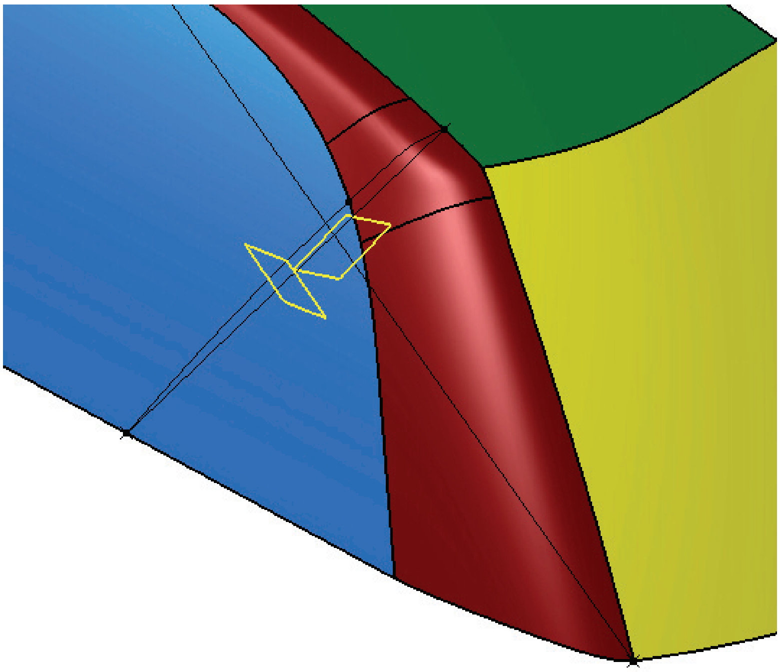


Fig. 7.19 The first normal plane is intersected with Class-A surfaces. The second secant results in the second normal plane.

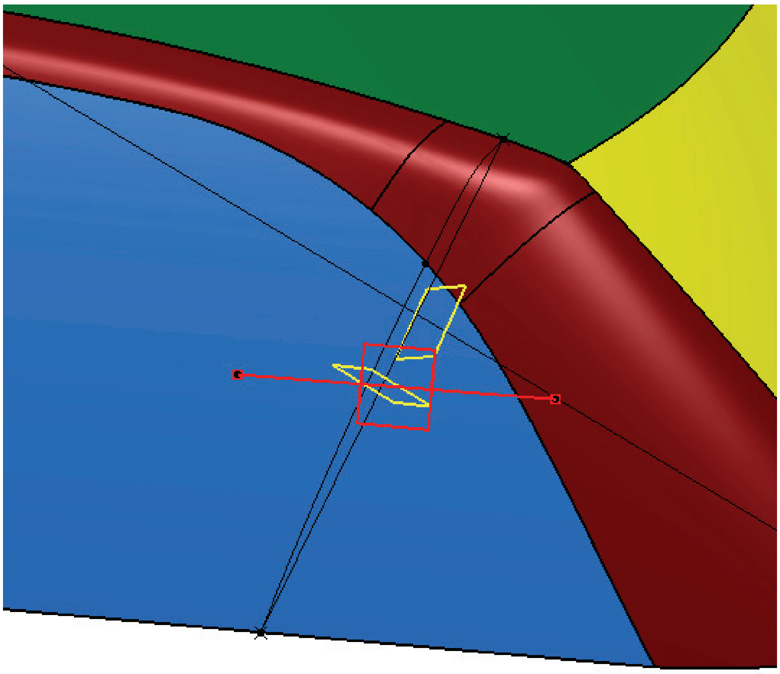


Fig. 7.20 The intersection line between the first and second normal plane defines tooling direction and plane

The intersection line of first and second normal plane is the tooling direction of the inner side panel, upper C-pillar. As the intersection line of two planes is infinite two interchangeable points are defined to allow an exchange of points for an alteration of the tooling direction according to tooling requirements in the later stage of the development. The two points are connected by a straight line which is defined to be the tooling direction. A normal plane to the tooling direction is the tooling plane used for the design of joint surfaces and approval of the prismatic surfaces (see figure 7.20).

7.5.3 Design of Profiles according to Tooling Requirements

Surfaces of an automotive body which do not strictly have to fulfil certain functions such as bedding functions do not have to be designed prismatic. For these surfaces the manufacturability is essential.

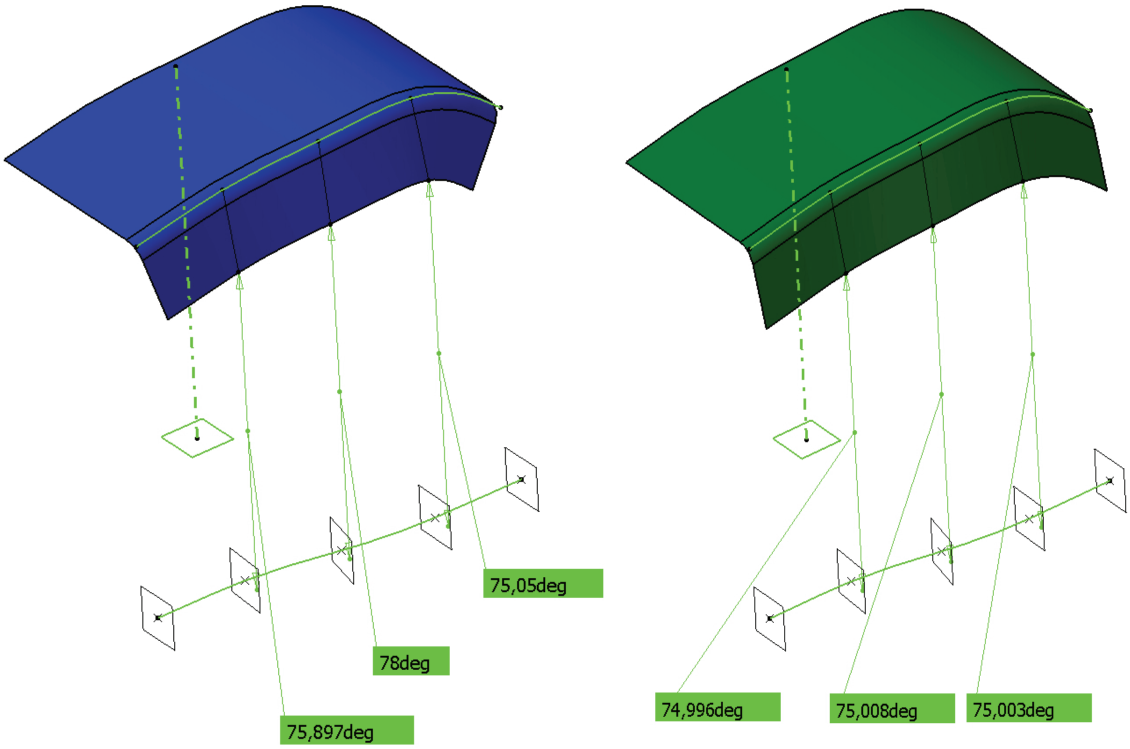


Fig. 7.21 Collar surface designed prismatic (left hand) and with constant release angle (right hand)

A comparison of two collar surfaces (see figure 7.21) shows that the collar surface designed according to the prismatic principle can be released from tool without any problems. In this example a release angle of 75° between flange and tooling plane (= 15° release angle between flange and tooling direction) does not fall below the expected angle of 75°. As long as the true most concept section controls the angles of bent portions which are much bigger than the minimum release angles (St 3°, Al 7°) the sloped surfaces can be designed prismatic without concerns.

As soon as it is necessary to design a surface with constant release angle or with changing release angle controlled by a law, other reference geometries have to be taken into account. For the design of a prismatic surface spine and guide curves are identical and the generating curve or profiles are constant.

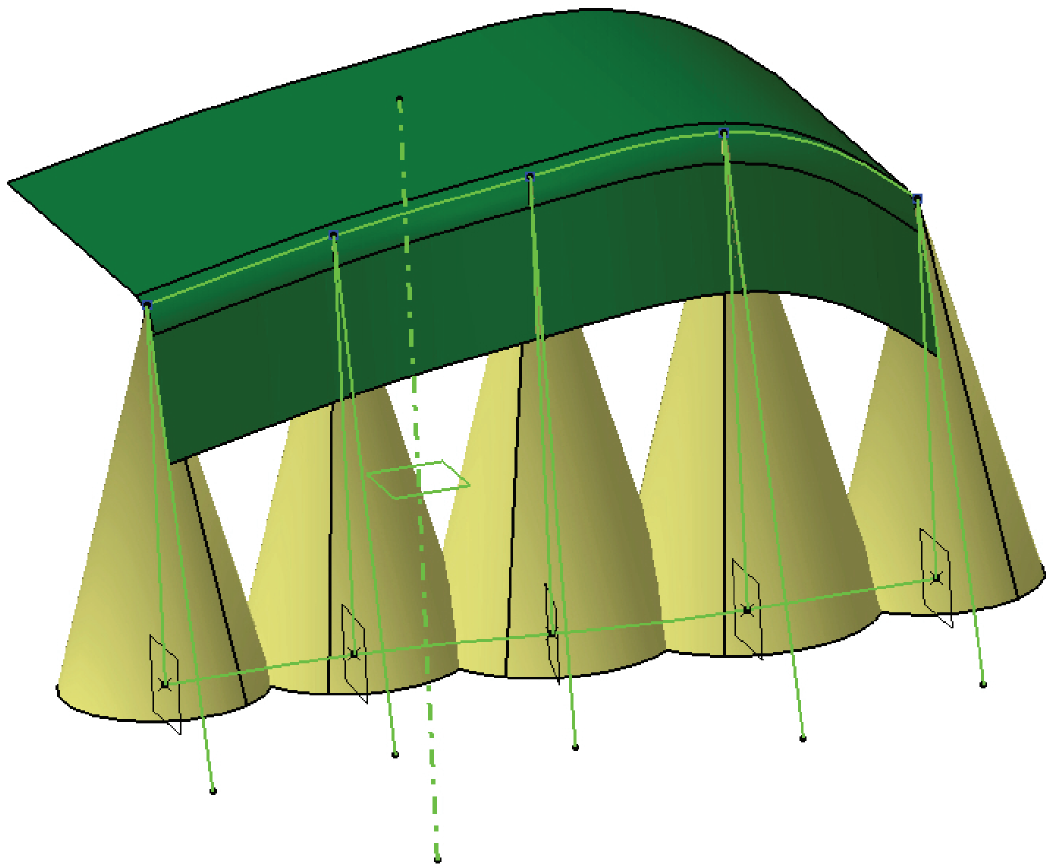


Fig. 7.22 A release cone perpendicular to the tooling plane moves along the 3D guide curve of the reference surface and generates a surface with constant release angle.

The reference surface in most cases is a secant surface (skewed surface) designed on the basis of another reference surface and the width of the prismatic profile. For a surface with constant release angle the reference surface is the tooling plane. The guide curve is an element of the reference surface while the curvature continuous spine curve is a projection of the guide curve onto the tooling plane. Under support of a cone with a point angle equivalent to the release angle which is moved with its tip along the 3D guide curve and is arranged perpendicular to the tooling plane the generation of a sloped surface with constant release angle can be easily understood (see figure 7.22).

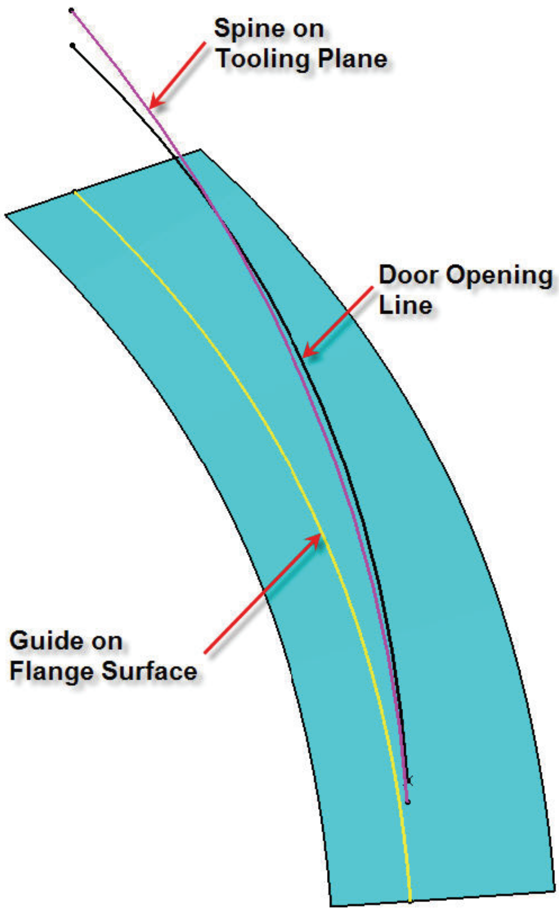


Fig. 7.23 Preparation of spine and guide curves for bent portions with constant release angles

Once the prismatic design of the door bedding, sidewall outer, C-pillar upper and the flanges of reinforcement C-pillar upper and sidewall inner C-pillar upper of the example C-pillar described above the further surfaces of door and window bedding are to be designed according to tooling requirements. These surfaces are the bent portions of reinforcement C-pillar and inner side panel, the part description and the tooling demands are defined according to the 3 series BMW.

Flange and recess surfaces of the outer bedding profile (outer side panel) as well as the flanges of the other two parts involved are designed prismatic. The bent portions of reinforcement C-pillar and side panel outer, upper C-pillar have to be designed with a constant release angle measured between surface band and tooling plane.

Therefore the opening line (in this case the door opening line) has to be projected onto the tooling planes of both parts to get spine curves for the surface bands under consideration of tooling requirements (see figure 7.23).

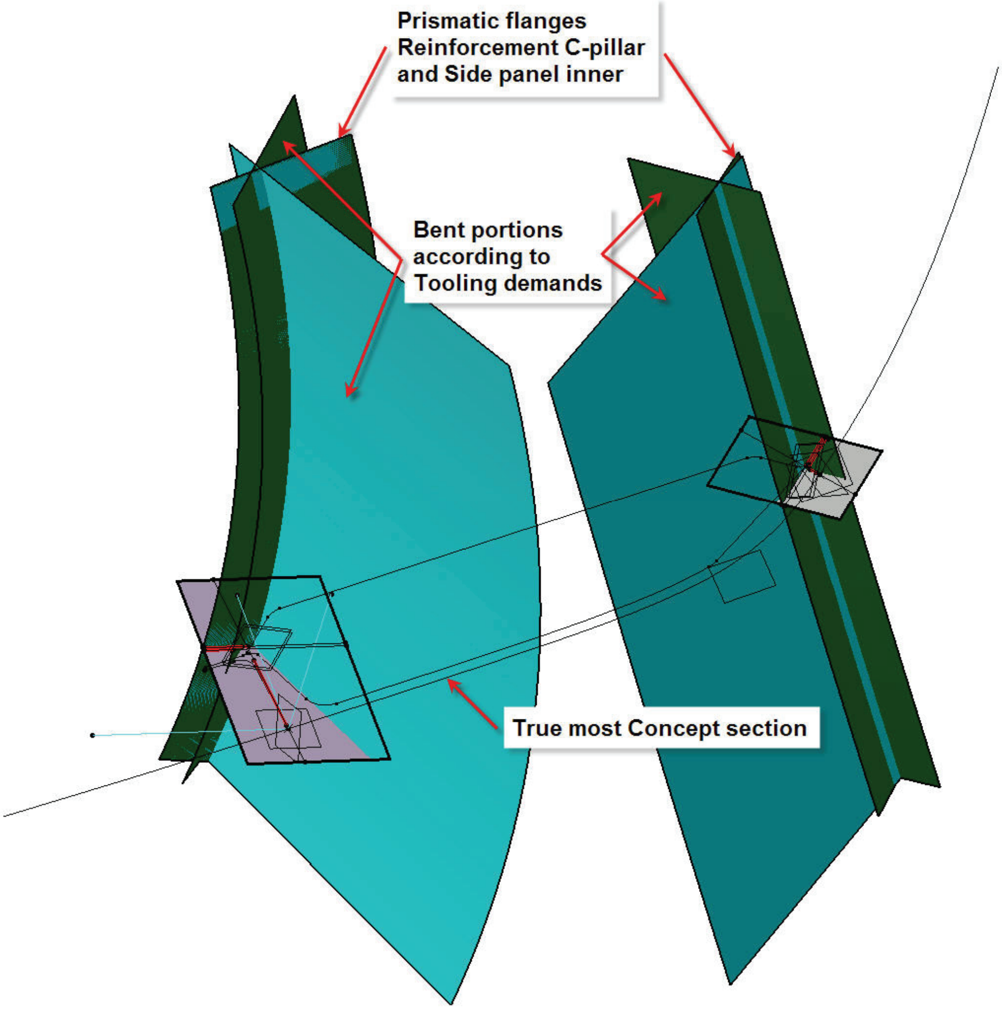


Fig. 7.24 The release angles of the bent portions are adjusted according to the demands of the true most concept section

The guide curve for the bent portion is the intersection curve of the prismatic flange and the bent portion and can be designed by projection and offset of existing curves. The spine curve of the releasable bent portion is the projected opening line. The minimum release angle for deep drawn steel shells is a predefined explicit parameter. The release angle used for the design of the two bent portions has to be adjusted according to the true most concept section. As spine and guide curves of these designs are not identical the surface bands of the two bent portions are not prismatic. That is to say that the angle between tooling plane and bent portion is constant but not the angle between bent portion and flange (see figure 7.24).

Once the flanges and bent portions of all three parts involved have been rounded the design of the door bedding has been completed. As the door bedding and window bedding are very similar the door bedding can be replicated and adjusted for the window bedding with definitions, directions and dimensions. All links of the replicated design have to be redefined. The design steps to complete the C-pillar and derive parts are similar to the description in section 7.2.4.

7.6 Methodology – Parametric associative Package Sections

This new approach closes a circle. In the past isolated package sections where and in some cases still are the basis for the composition and dimensioning of the true most concept section of a body zone (see section 7.2). The true most concept section is the prerequisite for the true radial compound sections which are necessary for the design of surface bands and profile surfaces. A package section of the completed body zone controls and/or documents certain specifications and/or legal requirements of the body zone. Its isolated copy could become prerequisite for the next car line project. The parametric associative package section is the reusable model of the body zone including the whole design approach which can be reused and adjusted as necessary.

In this important design application the use of update safe designs under support of the methodologies and principles described above is shown. The main idea is not to hand over isolated package sections (see figure 7.25) from car line to car line but to define 3D design zones including 2D package sections which can be automatically derived from the 3D approach once the design is completed.

In the concept development of a typical automotive body about 100 package-sections are developed and enhanced. Every package-section represents a zone or a sub-zone of the body (see figure 7.25). A package-section describes the composition and number of body parts involved in that zone. With the support of package-sections functions and legal requirements are verified and/or documented. In an early stage of concept development package-sections are often taken over from similar car lines of the OEM or his competitors. These sections often are distorted in the new design approach with respect to position and size. In the course of concept development, these sections are often updated to allow the effective verification and documentation of functions and legal requirements (see section 7.1). At the end of a car line project these package sections can be handed over to the next project.

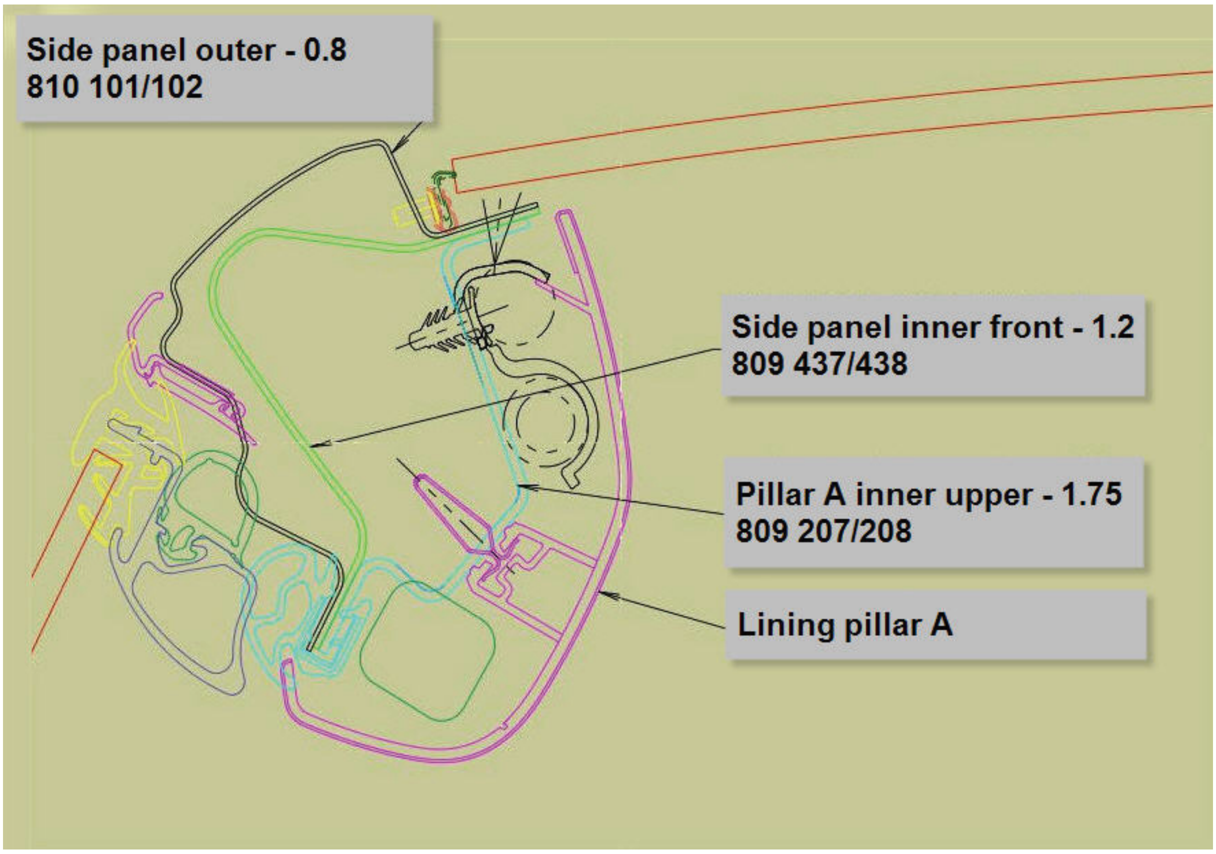


Fig. 7.25 Isolated Package section (true most radial position) of an A-pillar (AUDI)

Package sections are often neither true most nor true sections (see figure 7.26). Therefore these sections cannot be used for the design of surfaces and parts. In this chapter a methodology is described which combines development and enhancement of package-sections and the design of the zones of the automotive body. Every zone and associated package-section is defined in a continuously parametric associative configuration. This automated zone and section configuration can be reused from one development project to the next.

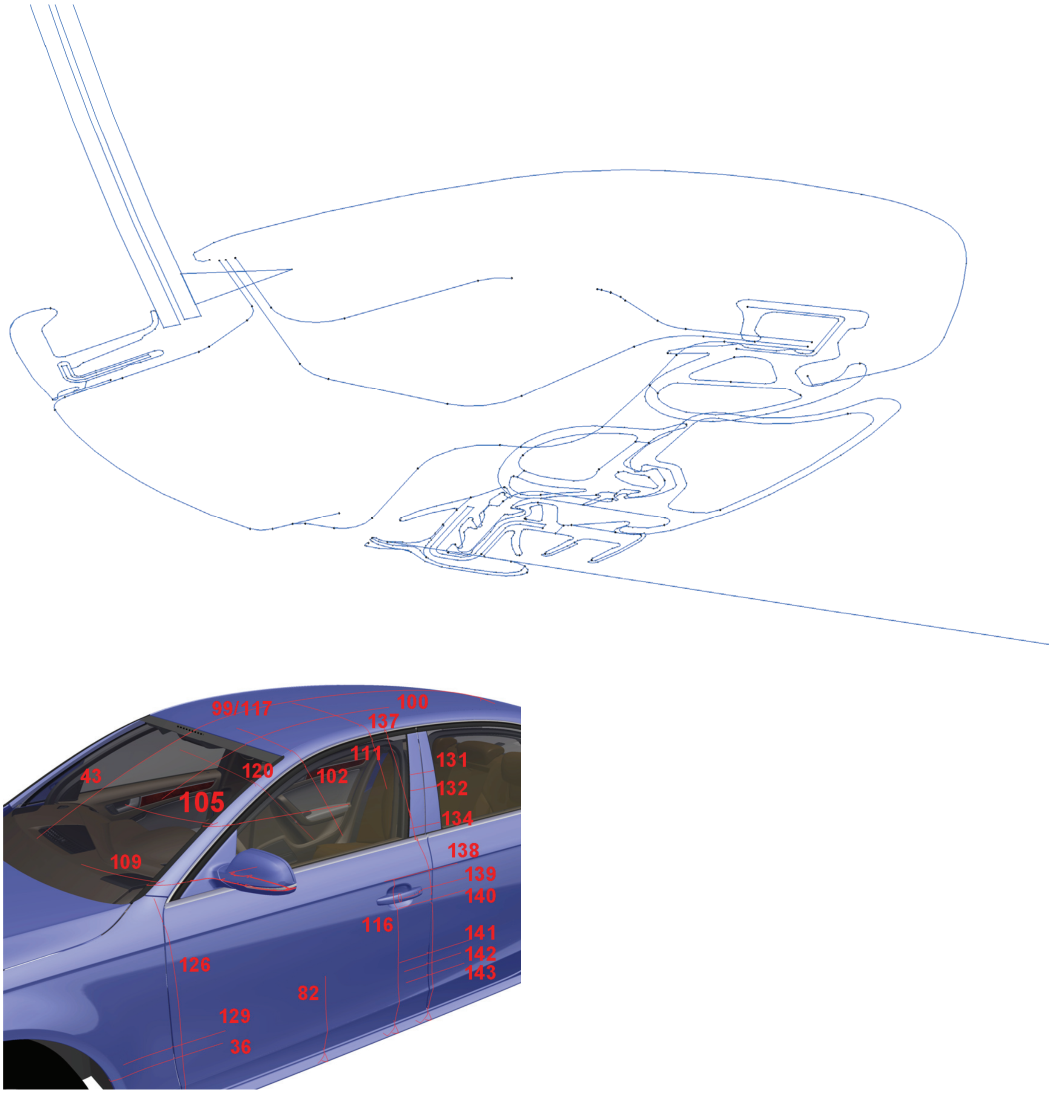


Fig. 7.26 Upper A-pillar package section 105 – section (top) and positioning (bottom - Nikol, AUDI)

According to (EG) 77/649 the package section of the upper A-pillar is a section on the height of the eye-points. For the design of parametric associative package sections a body segment with package sections can be derived from the workbench example "design zone upper A-pillar" (see figure 7.27). The segment is limited at the top and bottom by the two planes which are defined in EU directive (EG) 77/649 for the design of viewing beams. On the height of the eye points a third plane is defined for the approval according to the current legislation as well as to the former stronger legislation regarding ergonomics and geometry. See figure 7.27 shows that a binocular obstruction of 2.586° is achieved and the current legislation is fulfilled. But figure 7.28 shows that the real allowed binocular obstruction of 6° is exceeded by 2.176° and the concept design has to be adjusted. Still the most German OEMs use this former legislation. The binocular obstruction not only depends on the size of the cross-section of the A-pillar (true radial section) but also on the inclination and bending of the A-pillar as well as on the distance between eye points and pillar.

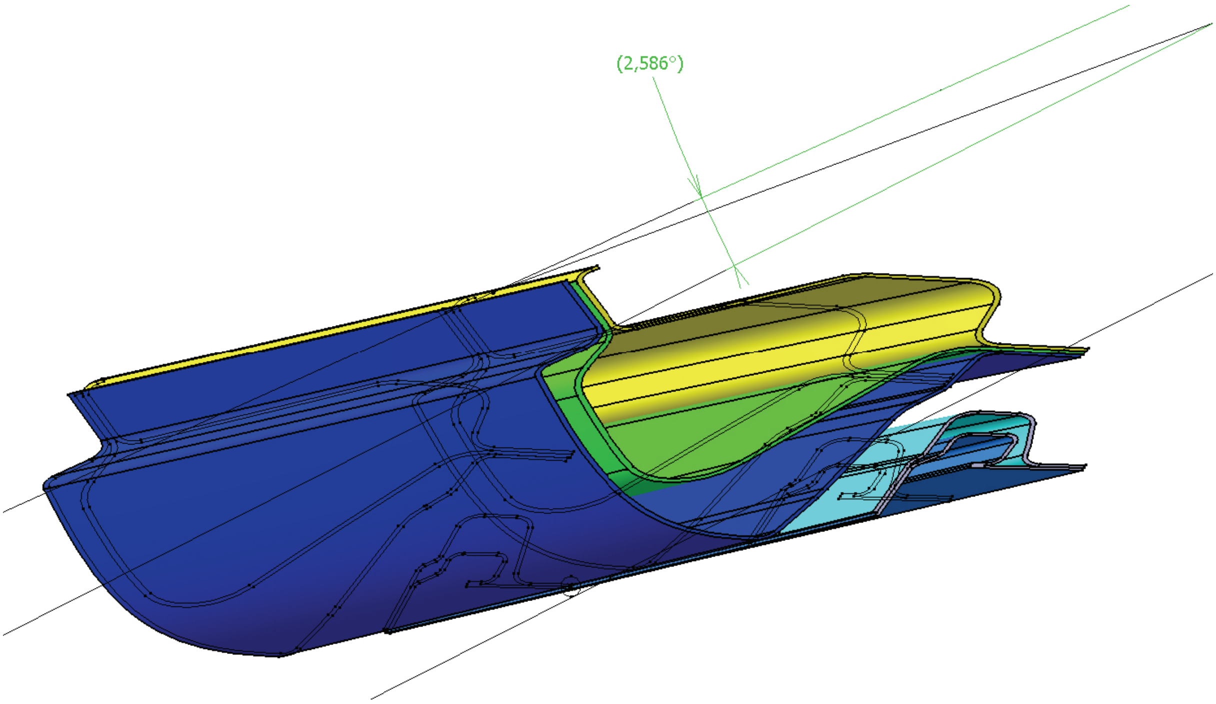


Fig. 7.27 Parametric re-usable 3D approach and package-section with current binocular obstruction according to (EG) 77/649

The new parametric associative design approach described allows the exchange of reference geometries such as Class A surface, spine curves etc. changes of the seating position of the driver or of the dimensions of the section or alterations of explicit and implicit parameters at any time.

As shown in this chapter the update safe development of a body zone combined with the derivation of the parametric associative package-section consequently allows the automatic adaptation of new requirements and approval of e.g. the binocular obstruction. Compatibility for new references and flexibility for new requirements results in a high degree of re-usability of the CAD-model.

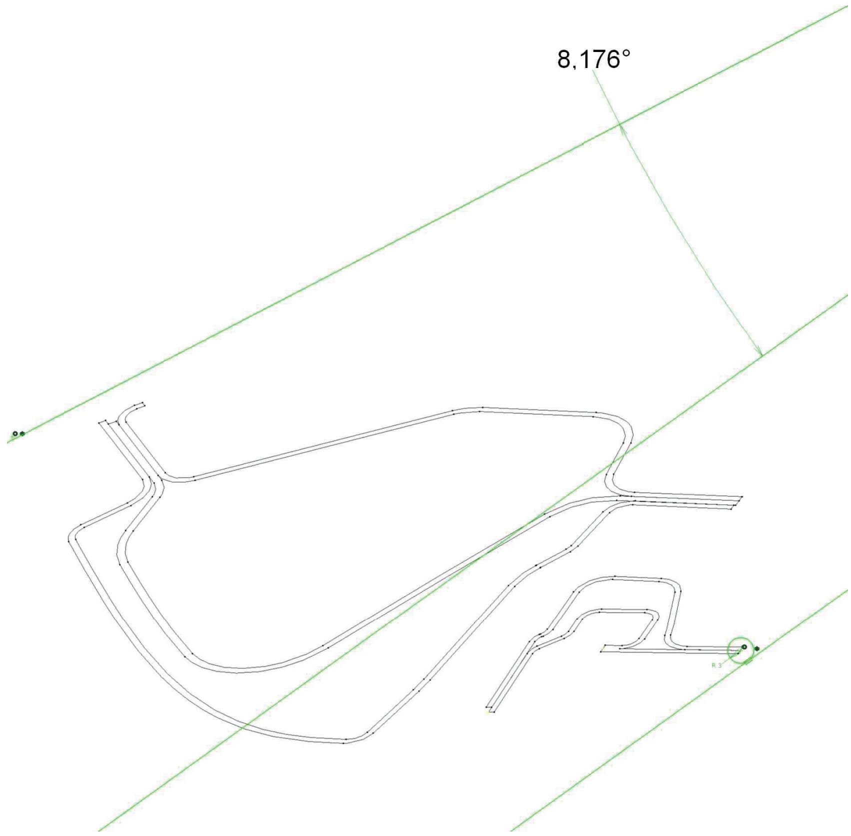


Fig. 7.28 Parametric associative package section with current binocular obstruction according to OEM specification

7.7 Evaluation of Methodologies and Principles

To improve the seven methodologies and principles for the design of prismatic surfaces described above three automotive body zones (work benches) have been allocated and designed. These are the zones of a) a rear roof rail with prismatic flanges at its front plus prismatic window bedding, b) a lateral roof rail with prismatic roof and door bedding, and c) the zone of an upper C-pillar with prismatic door and window bedding. With the three test benches design environments are available with two prismatic profile areas each and a different depth of complexity.

These work benches have been designed seven times each (21 CAD models) to evaluate the methodologies and principles under different conditions and to achieve conclusive evaluations for the methods.

7.7.1 Description of the three Work Benches

7.7.1.1 Roof Rail rear

Along the intersection curve between the Class A surfaces of roof and rear wall the roof rail rear is positioned. The sections necessary for the product evolution process (PEP) such as package section, concept section and true compound radial sections for derivation of prismatic surface areas are identical. The rear roof rail is symmetrical according to the vehicle centre plane (ZX plane) and all the sections are positioned on the symmetry plane ZX (see figure 7.29).

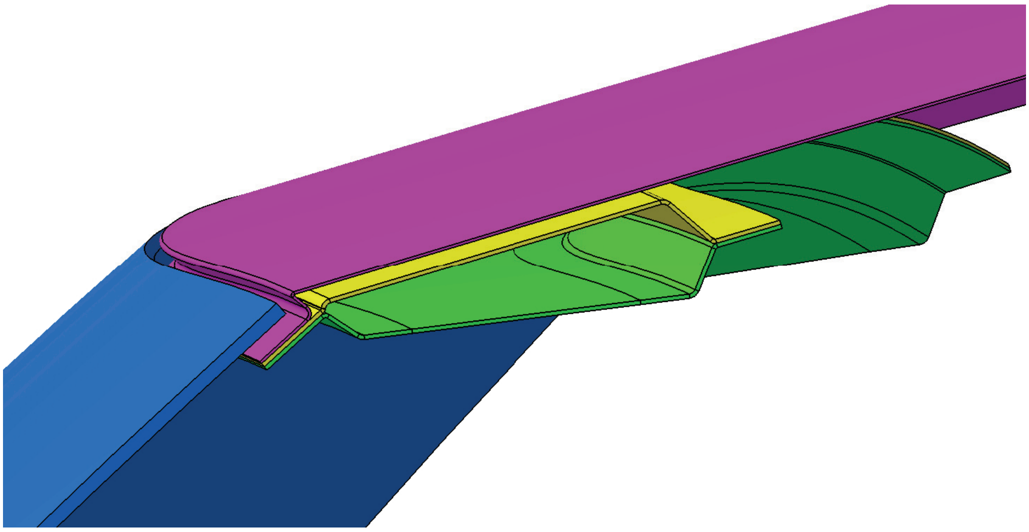


Fig. 7.29 Sectional view Y0 rear roof rail

The roof rail is defined from requirements in horizontal vision, head clearance rear and stiffness of body structure. The roof rail consists of a lower and an upper shell connected along two prismatic spot welding flange areas. Along the rear flange the roof is spot welded to the roof rail and the rear window is bonded to the spot welding flange.

7.7.1.2 Roof Rail side

Perpendicular to the intersection curve between the Class-A surfaces of roof and side wall the concept section for the definition of the roof rail side is defined above the rear door (see figure 7.30). The positions of the true radial concept sections for the prismatic roof and door beddings are defined perpendicular to the door opening line and the water management line roof. The planes are defined at the intersection points of the concept plane and the two spine and guide curves. Shape and position of the roof rail define two compound radial planes which are similar to the radial plane of the concept section. From the true sections surface bands are derived which are the basis for the definition of the junction between roof rail rear, roof rail side and C-pillar upper and the determination of body part sections.

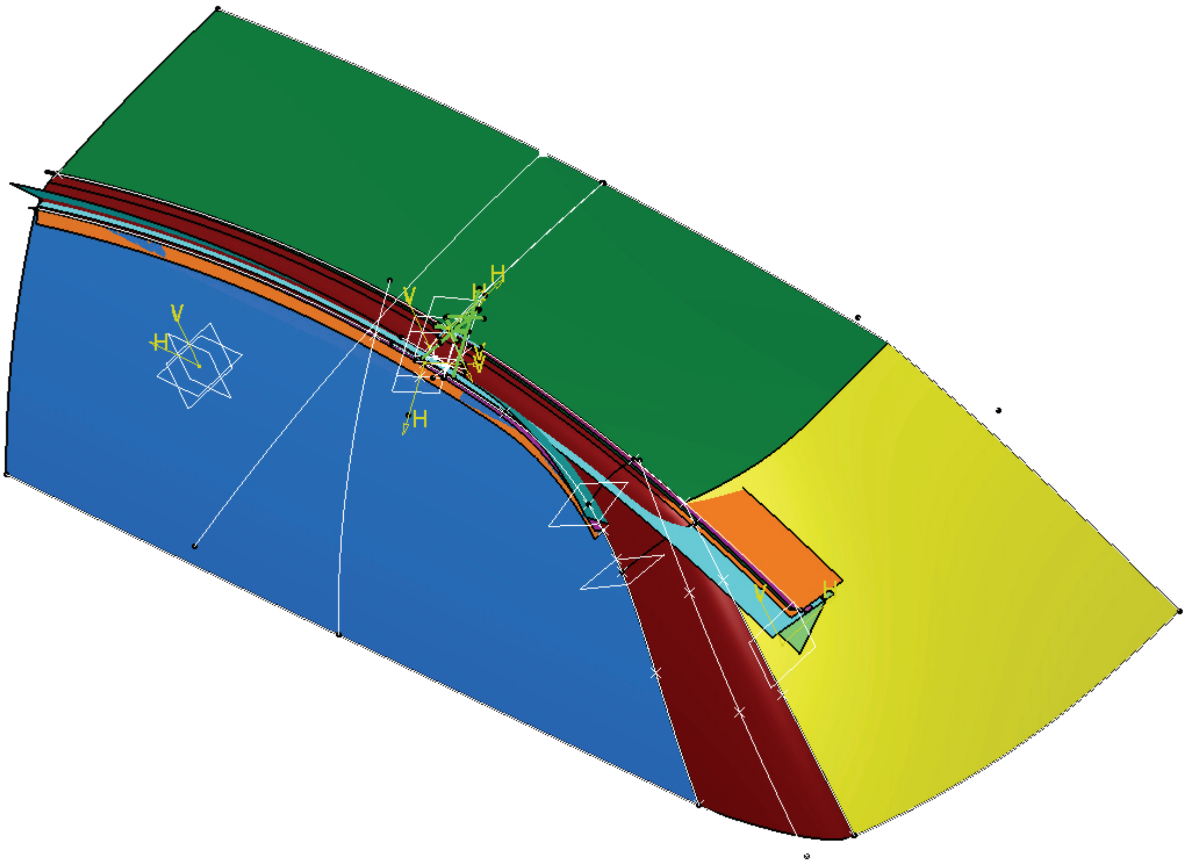


Fig. 7.30 Roof rail side

The roof rail side is defined according to specifications of side impact, body stiffness, head clearance and access comfort. The prismatic bedding of the roof shows a step between roof surface and sidewall to lead water along the step into the rear area of the car. Both parts are laser soldered along this step. Behind the B-pillar the roof rail side is build from side panel outer, reinforcement C-pillar upper and side panel inner, C-pillar upper. The parts are spot welded to each other along two prismatic bedding areas.

7.7.1.3 C-Pillar upper

In the middle of the intersection curve between the Class A surfaces of rear wall and sidewall the true most radial concept section for the design of the upper C-pillar is defined. The planes for the true compound radial sections for the development of the prismatic door and window beddings are defined perpendicular to door and window opening lines. The planes are defined at the intersection points of the concept plane and the two spine and guide curves. Due to the shape and position of the C-pillar the positions of the concept plane and the two normal planes for the true sections are very different (see figure 7.31).

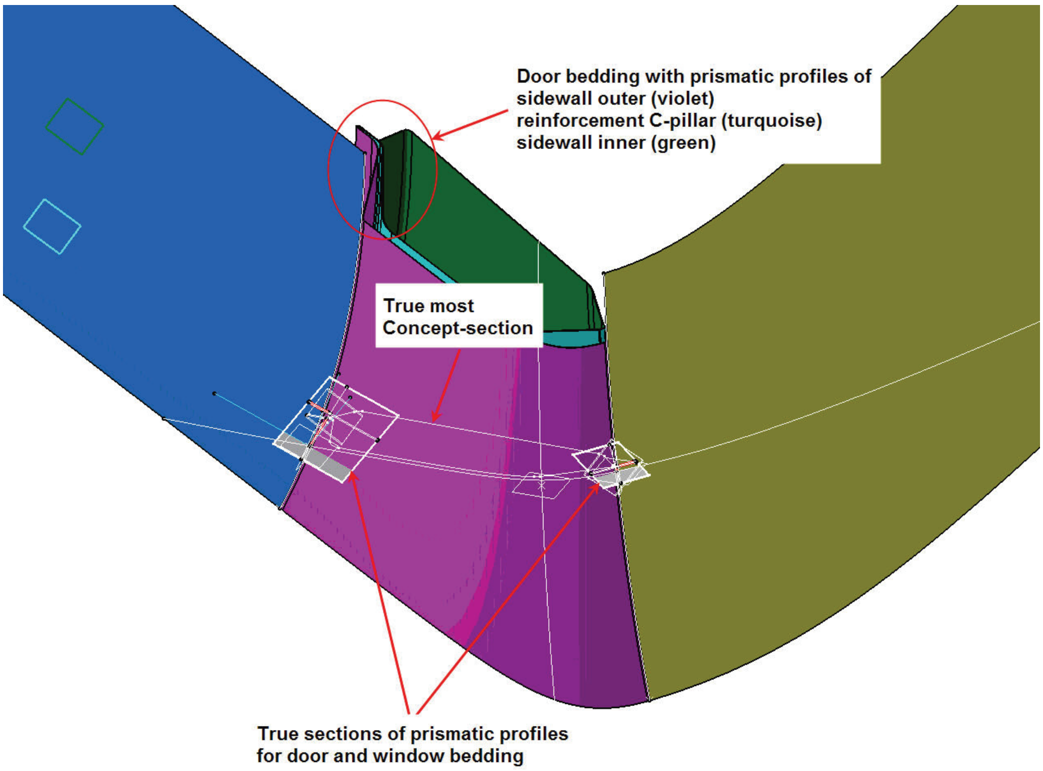


Fig. 7.31 C-pillar upper with true most radial concept section and two true compound radial sections of prismatic bedding areas

From the true sections surface bands are derived which are used for the design of the junctions C-pillar/roof rail and C-pillar/wheel arch upper or the determination of body part sections using surface bands from other body zones.

The C-pillar upper is build from side panel outer, reinforcement C-pillar and side panel inner C-pillar upper. The three parts are spot welded to each other along the two bedding flanges.

7.7.2 Scenarios of Variations

In the completed 21 CAD-models (3 work benches and 7 different methods and principles) different variations of references and explicit parameters are conducted to check and compare their update-stability (see tables 7.1 to 7.3). Besides update stability the update time was measured. Variations are carried out on the different characteristics of the segments of profiles itself as well as on the reference geometries. The scope of variations includes realistically once as well as overdrawn.

7.7.3 Validation of Design Methods and Principles

First of all it is necessary to check whether all CAD-models are designed with zero defects and whether the hierarchical tree is structured in clearly arranged design sequences. All methods and principles are reviewed and compared according to the same evaluation scheme. The design effort and number of features until completion of the CAD-model and the analysis of update stability under different variations are considered. The evaluation criteria are classified in four categories: **ease of use, traceability, update stability and effort reduction by replication.**

The weighting of the evaluation criteria is carried out in a matrix according to Kepner, Tregoe, 1965 (see table 7.4). For the utility analysis according to Kepner, Tregoe, 1965 (see table 7.5) the single criteria are marked from 1 to 6, at this mark 1 stands for the best and mark 6 stands for the worst rating. The marks are multiplied with the weighting factors of the first table. The total of the weighted marks decides on the ranking of the different design methods. The update times measured for all variations are not taken into further account as there are only small differences.

According to the criteria **ease of use** the complexity of the profiles, the effort to edit the extension of the segments and the design features is taken into account. If a simple profile is to be designed that is to say a small number of straight segments with a small number of segment extensions the ease of use must be quick and easy. This is often the case when a rough layout in the early stage of concept development is elaborated. In later stages of concept development the designs and the profiles are more detailed and more complex. In these stages the number of segments and segment extensions gets less important. More important for these stages is the effort for a well structured design composition of the particular method, the traceability and the update stability.

Scenario A (Roof rail rear)	**Old**	**New**
Reference geometries		
Roof: 1700X (roof height)	1300mm	1350mm
Roof: 0Y (roof height front)	1300mm	1200mm
Rear Wall Upper Class A: 800Z (Position rear window)	3370mm	3450mm
Rear Wall Upper Class A: 0Y (Position rear window)	38°	45°
Explicit Parameters (Profile)		
Length Flange Window Bedding Outer	20mm	18mm
Depth Recess Window Bedding Outer	9mm	10mm
Angle Flange Recess Window Bedding Roof	80°	110°
Angle Window Bedding Roof Rail Upper Shell	140°	110°
Parallel Roof	30mm	40mm
Length Flange Prismatic Area Roof Rail Upper Lower Shell	21,5mm	25mm
Bent Portion Prismatic Area Roof Rail Upper Shell	15mm	12mm
Bent Portion Prismatic Area Roof Rail Lower Shell	12mm	15mm

Table 7.1 Variation roof rail rear (Scenario A)

Scenario B (Roof rail side)	**Old**	**New**
Reference geometries		
Side Wall Class A: 800Z (Vehicle width)	850mm	900mm
Side Wall Class A: 1750X (Position side wall)	1150mm	1250mm
Roof Class A: 1750X (Roof height)	1350mm	1250mm
Roof Class A: 0Y (Roof height rear)	1300mm	1600mm

Explicit Parameters (Profile)		
Angle Door Bedding Roof Rail Side Wall Inner	130°	160°
Angle Door Bedding Roof Rail Reinforcement	110°	130°
Length Flange Door Bedding Side Wall Inner	22mm	25mm
Sheet Thickness Reinforcement C-Pillar	1,2mm	1,6mm
Distance Roof Bedding Reinforcement	18mm	20mm
Length Prismatic Profile Roof Bedding Side Wall Inner	20mm	18mm
Angle Roof Bedding Side Wall Outer	132°	150°

Table 7.2 Variation roof rail side (Scenario B)

Scenario C (C-pillar)	**Old**	**New**
Reference geometries		
Rear Wall Upper Class A: 800Z (Position rear window)	3370mm	3450mm
Rear Wall Upper Class A: 0Z (Position rear window)	45°	50°
Side Wall Class A: 800Z (Vehicle width)	850mm	900mm
Side Wall Class A: 1750X (Position side wall)	220mm	200mm
Explicit Parameters (Profile)		
Length Flange Door Bedding Side Wall Outer	25mm	27mm
Length Flange Door Bedding Reinforcement	27mm	29mm
Length Flange Door Bedding Side Wall Inner	22mm	25mm
Sheet Thickness Reinforcement C-Pillar	1,2mm	1,6mm

Table 7.3 Variation C-Pillar upper (Scenario C)

For the evaluation of the segment extensions according to the single principles the design effort, the type of auxiliary geometry and the traceability of the result is examined. One important element is the workbench *Sketcher*. If the extension design is carried out in the *Sketcher* incorrect, results may appear when geometrical constraints are used. When an update safe CAD-model is expected geometrical constraints must be defined carefully. Is the segment extension carried out in the 3D modelling space the use of Boundary Representing Elements (BReps) must strictly be avoided.

No.	Weighting of evaluation criteria according to Kepner-Tregoe	1 Ease of use for simply profiles	2 Ease of use for complex profiles	3 Ease of use for segment extension	4 Ease of use - number of design steps	5 Tracebility for third parties	6 Tracebility - clear arrangement structural tree	7 Update stability - Geometrical constraints	8 Update stability - Effort to edit tree or features	9 Update stability when references are altered	10 Effort reduction by replication	Number of points	Ranking	Weighting [%]
No.	**Evaluation criteria**													
	Ease of use													
1	Ease of use for simply profiles	1	0	0	0	0	0	0	0	0	0	1	10	1,82
2	Ease of use for complex profiles	1	1	0	0	0	0	0	0	0	0	2	9	3,64
3	Ease of use for segment extension	1	1	1	1	0	1	0	0	0	1	6	5	10,91
4	Ease of use - number of design steps	1	1	0	1	0	1	0	0	0	1	5	6	9,09
	Traceability													
5	Tracebility for third parties	1	1	1	1	1	1	0	0	0	1	7	4	12,73
6	Tracebility - clear arrangement structural tree	1	1	0	0	0	1	0	0	0	1	4	7	7,27
	Update stability													
7	Update stability - Geometrical constraints	1	1	1	1	1	1	1	0	0	1	8	3	14,55
8	Update stability - Effort to edit tree or features	1	1	1	1	1	1	1	1	1	1	10	1	18,18
9	Update stability when references are altered	1	1	1	1	1	1	1	0	1	1	9	2	16,36
10	**Effort reduction by replication**	1	1	0	0	0	0	0	0	0	1	3	8	5,45
	Score											55		100

Table 7.4 Weighting of evaluation criteria acc. to Kepner, Tregoe, 1965

The number of design steps for the definition of the profile, the segment extension and derivation of surface bands is another element. The proceedings are different from principle to principle. When the true section and the auxiliary geometry for segment extension are defined in workbench *Sketcher* the design effort can be kept low. If the true section is designed in 3D modelling space, it is necessary to design a section for every segment. When the surface bands are developed without a true section, a control section for every surface band has to be designed and compared with the concept section.

The **traceability** is the criterion for the clear arrangement of the design model. For third parties every single design step must be traceable and the design features must be legible. This allows the easy and time-saving reuse or completion of the CAD-model. When a true section is designed the traceability of the true section including the auxiliary geometry for segment extension is controlled. Too much auxiliary geometry often results in an intransparent section. Another important issue is the clear arrangement of the hierarchical tree. The tree should reflect the systematic design approach. Every single design step should be legible and editable without long winding selections of hierarchies.

The most important criterion for the evaluation of parametric associative design criteria is the update stability. Design variations and changes of parameters very often take place in the concept phase. As parametric associative design delivers the most benefit in this development phase the update stability is of paramount importance. One bad influence is in the use of geometrical constraints in the *Sketcher*. If the auxiliary geometry for the extension of segments is defined in workbench *Sketcher* and geometrical constraints are used, the geometrical constraints must often be edited after the update. For every design principle the effort for editing is rated to a complete update without failures. The update time including editing operations often depends on the number of sketches and design features which have to be chosen and opened for editing.

The design work is easier when all important parameters for the profiles are visualised in a true section which is not crowded with distracting auxiliary geometry. In addition a very good section shows the influence of parameter variations on the whole profile. It is important to safeguard the consistency of the section by *Sketch Analysis.* On the other hand this analysis takes extra time when a section is developed or edited.

The best update result is when the designed method can be updated without any failure. The shape and position of a body part will vary during the course of development process. Therefore it is particularly important that a design method is updating stable when reference geometries are altered or exchanged.

The **replication** of design steps can reduce the design effort. For the replication the design steps must be sorted in a *Geometrical Set*. The advantage of replication occurs as soon as the effort for sorting and editing of nomenclature and links is smaller than the manual design of several similar design steps.

7.7.4 Discussion of Results of Utility Analysis acc. to Kepner, Tregoe, 1965

7.7.4.1 True Section Method - 3D only

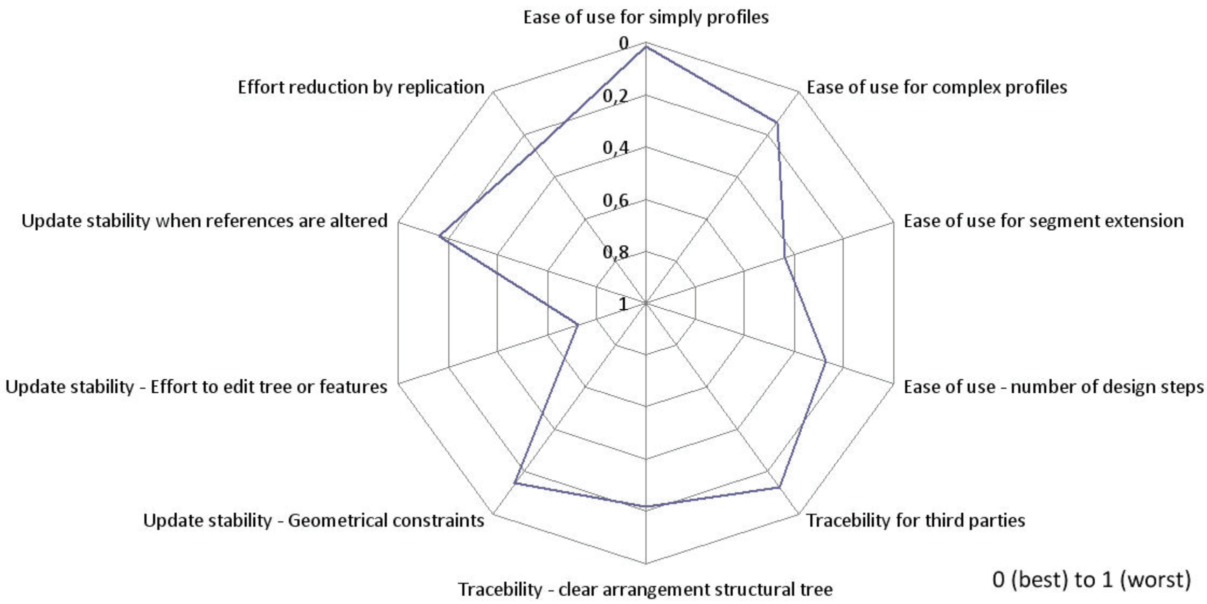


Fig. 7.32 Validation of True section method – 3D only

	PRINCIPLE		I True section methode - 3D only	II True section methode - extension by	III True section methode - boundary circle	IV True section methode- boundary box	V True section methode - boundary box (surface)	VI True section methode - extrapolation 3D	VII Surface band methode
No.	Evaluation criteria	Weighting	Rating	Rating	Rating	Rating	Rating	Rating	Rating
	Ease of use								
1	Ease of use for simply profiles	1,82%	1	2	2	2	2	1	1
2	Ease of use for complex profiles	3,64%	4	3	5	2	1	2	4
3	Ease of use for segment extension	10,91%	4	5	5	3	2	3	3
4	Ease of use - number of design steps	9,09%	3	2	1	1	2	2	5
	Tracebility								
5	Tracebility for third parties	12,73%	1	2	5	1	1	1	2
6	Tracebility - clear arrangement structural tree	7,27%	3	1	2	1	3	2	4
	Update stability								
7	Update stability - Geometrical constraints	14,55%	1	4	5	4	1	2	1
8	Update stability - Effort to edit tree or features	18,18%	4	2	2	1	1	3	5
9	Update stability when references are altered	16,36%	1	4	5	3	1	2	1
10	**Effort reduction by replication**	5,45%	5	1	5	2	1	5	5
	Final score	**100,00%**	**2,53**	**2,85**	**3,82**	**2,09**	**1,36**	**2,31**	**2,98**

Rating 1 (best) to 6 (worst)

Table 7.5 Utility analysis of design methods acc. to Kepner, Tregoe, 1965

The design of the true section and the segment extension in 3D modelling space is a traceable and updates save method. For every single segment a true section is designed. The hierarchical tree is long but legible even for third parties. Every single design step can be followed directly. The exclusive work in 3D avoids the use of unsafe geometrical constraints in workbench *Sketcher*. As the segment extension takes place by extrapolation it is important not to use any BReps.

The decision for this method depends on the complexity of the profiles. As soon as the profiles get more complex this method gets confusing because all interdependent design steps must be controlled in curve and surface directions, positions and expansions and be compared with the concept section. A simplification by replication is not suitable. As the single design steps are often different, the effort for replication is higher than the repeated design steps. The high number of design steps affects the effort when references have to be edited. Even if every segment design is directly legible in the hierarchical tree every single segment must be selected and edited in case of extensive variations.

With an overall score of 2.53 (1.36 is the score of the best design method) this method belongs to the centre span (2.59) of the methods evaluated (see figure 7.32).

7.7.4.2 True Section Method – Extension by Dimensioning

The method to extend the segments in workbench *Sketcher* by dimensions is easy to understand for third parties. The hierarchical tree keeps short and legible. The traceability of the sketch keeps obtained even when the segments are extended. This is essential even for complex profiles.

All segment extensions are performed in the true section which saves design time. The replication of the completed design of the first bedding for the development of the second bedding is another time saving opportunity. As long as the profiles are simple this method is quite update safe. As the auxiliary points for the segment extensions are positioned by a geometrical constraint, a dimension editing failure may occur. All geometrical constraints have to be defined carefully.

When curved segments are defined by three points are involved the segments have often to be redesigned after the update cycle because the composition of curve defining points get lost. With this method the editing effort increases with the complexity of the profiles.

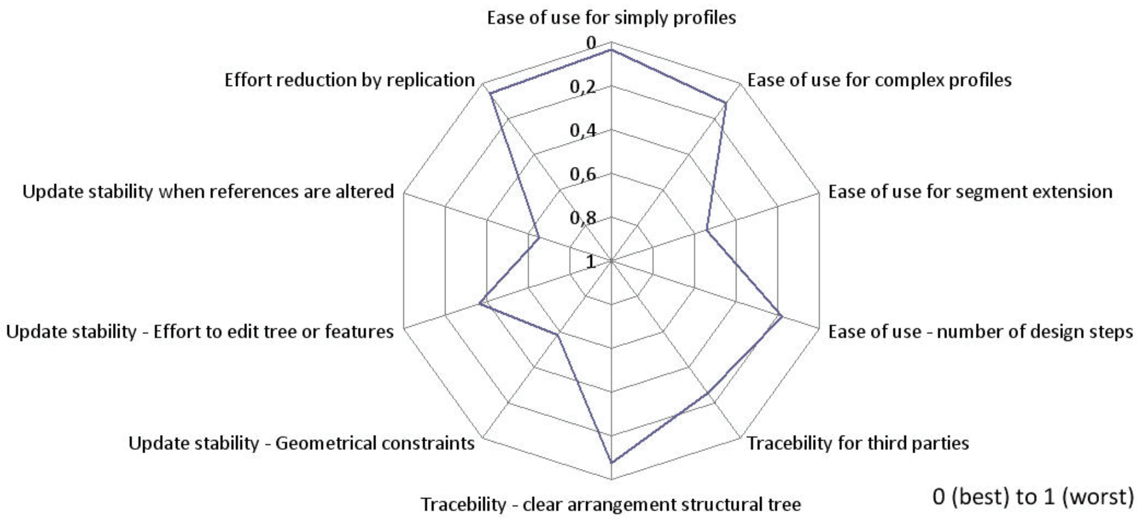


Fig. 7.33 Validation of True section method – Extension by dimensioning

With an overall score of 2.85 (1.36 is the score of the best design method) this method is below the centre span (2.59) of the methods evaluated (see figure 7.33).

7.7.4.3 True Section Method - Boundary Circle

The design of a true section and extension of its segments under support of auxiliary circles is an easy to understand design principle. For simple profiles this principle is time saving and legible. Design and extension work in the true section keeps the hierarchical tree short and traceable. As soon as profiles get more complex the number of auxiliary circles and geometrical constraints increase, hence making the sketch intransparent and leading to editing mistakes after updates.

The replication of the completed design of the first bedding for the development of the second bedding is a time saving opportunity.

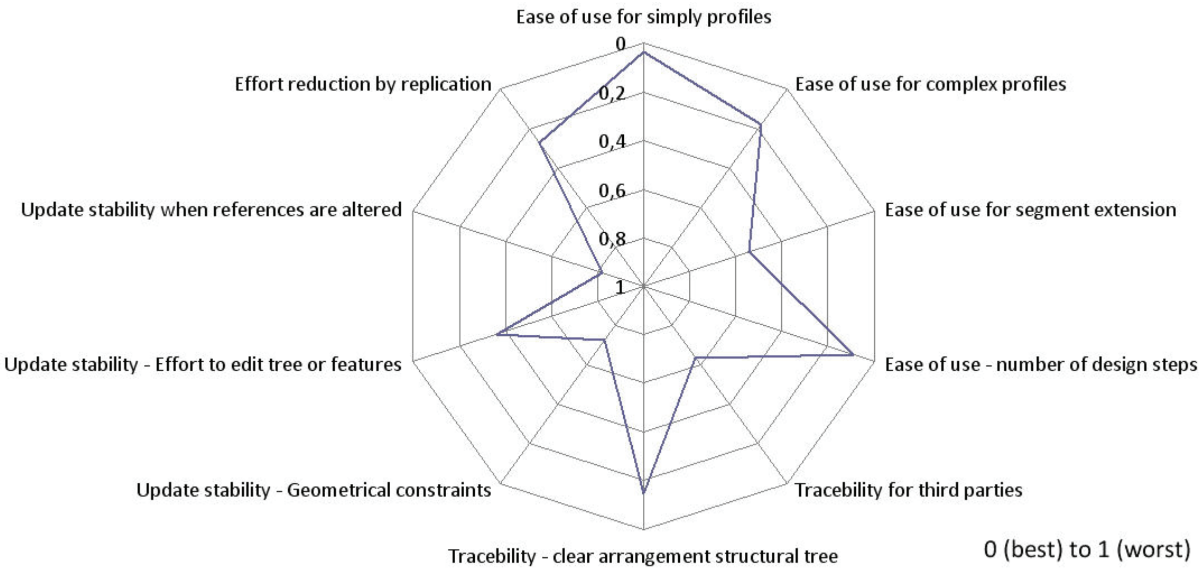


Fig. 7.34 Validation of True section method - Boundary circle

With an overall score of 3.82 (1.36 is the score of the best design method) this method has the worst rating of the methods evaluated (see figure 7.34).

7.7.4.4 True Section Method - Boundary Box (Wire Frame)

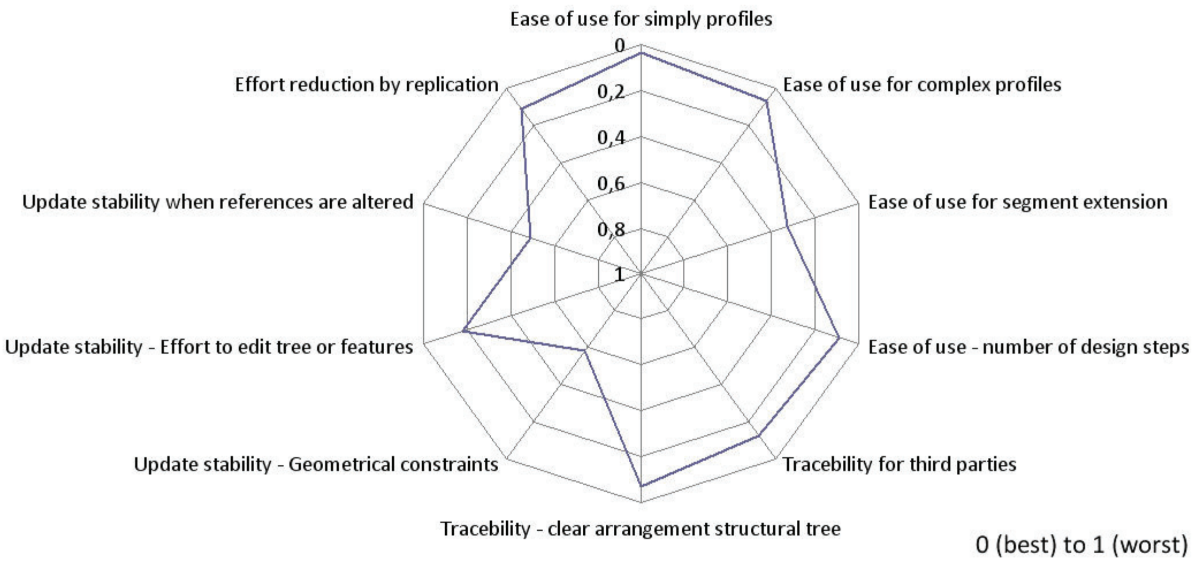


Fig. 7.35 Validation True section method - Boundary box (wire frame)

The design principle to extend the segments of a section in the workbench *Sketcher* under support of one auxiliary rectangle is suitable for straight and curved segments.

In contradiction to the boundary circle principle the single rectangular auxiliary geometry keeps the section legible even when the segments are extended. The extension of the segments in the *Sketcher* keeps the hierarchical tree short and traceable. The definition of geometrical constraints and the design and extension of curved segments needs special attention. The references in the short hierarchical tree and the legible sketch can easily be edited. The replication of the completed design of the first bedding for the development of the second bedding is a time saving opportunity.

With an overall score of 2.09 (1.36 is the score of the best design method) this method has the second best rating of the methods evaluated (see figure 7.35).

7.7.4.5 True Section Method - Boundary Box (Surface)

This principle with its auxiliary geometry developed in workbench *Sketcher* and the extension of segments in 3D is only suitable for straight segments. Curved segments must be extended under support of the wire frame principle. The entire method from sketch to surface bands is applicable very good even for complex profiles. With the extension design in 3D the hierarchical tree gets very long. The method allows a good transparency as the design steps for every segment and its related surface band is sorted in a *Geometrical Set*. The true section is kept untouched by the extension principle. The section can be supplemented with sheet thickness under support of offset curves and radii (manager section) without interference of the extension principle. Failures by using geometrical constraints can be avoided. The only disadvantage of the method is that curved segments must be extended in workbench *Sketcher*. Apart from that there are no problems with updates and editing of references. At a first glance the high number of design steps may be irritating. But extension in 3D and design of related surface bands can easily be achieved by replication of the first design which reduces the design effort.

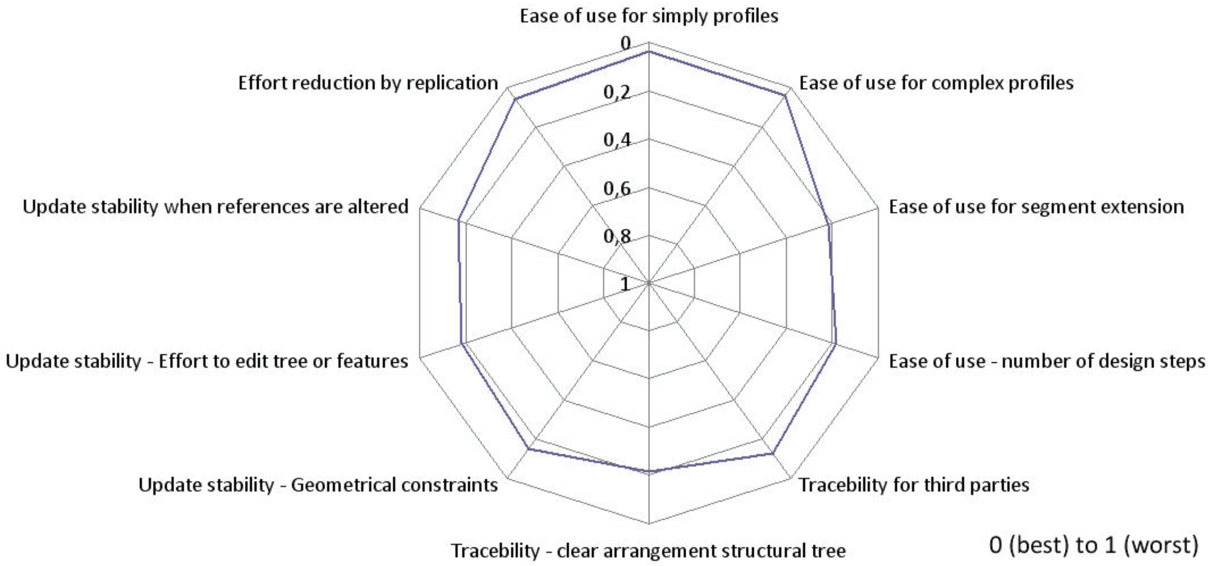


Fig. 7.36 Validation True section method - Boundary box (surface)

With its overall score of 1.36 this method is the best of all methods evaluated (see figure 7.36).

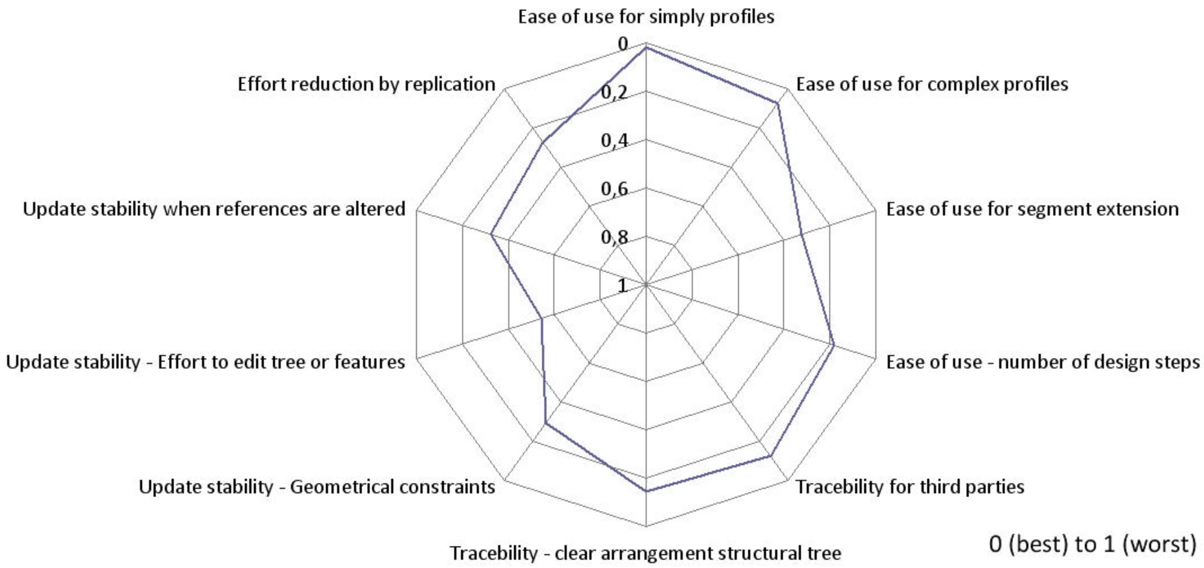


Fig. 7.37 Validation True section method - Extrapolation 3D

7.7.4.6 True Section Method - Extrapolation 3D

The design of a true section in workbench *Sketcher* and the segment extension in 3D by extrapolation is suitable for all kind of profiles. Auxiliary geometries for the extension of the segments are points already existing in the *Sketcher* which have to be diverted from the *Sketcher* into 3D. Afterwards the segments are extended in 3D by extrapolation under support of these points. The true section is completely untouched by the extension principle. The section can be supplemented with sheet thickness under support of offset curves and radii (manager section) without interference of the extension principle. The hierarchical tree is very long but still very good traceable as the number of design steps for each segment and related surface band is small.

The designer must be carefully not to use any BReps for the extrapolation of segments. Only existing start and end points diverted from the *Sketcher* should be used for extrapolation. In this method only straight lines and segments of a circle should be extended by extrapolation. The evaluation has shown that the extrapolation of splines results in follow up failures in the surface design developed from the extrapolated segment. For the extension of segments no geometrical constraints are necessary. The small number of the geometrical constraints for the design of the section makes the method very update stable. The only problems are curved segments. Curved segments could not automatically be updated as the composition of curve designing points gets lost. Points and curves may have to be redesigned.

With an overall score of 2.31 (1.36 is the score of the best design method) this method has the third best rating of the methods evaluated (see figure 7.37).

7.7.4.7 Surface Band Method

For simple profiles the surface band method can be used quickly and easily as the complete design takes place in 3D. The control of the curves and surfaces directions as well as the positions and expansions compared with the concept section to avoid failures in the design can be done quickly. Another advantage of the method is the exclusive design in 3D which avoids the use of geometrical constraints in workbench *Sketcher* and allows high update stability. The single design steps of simple profiles can be easily edited.

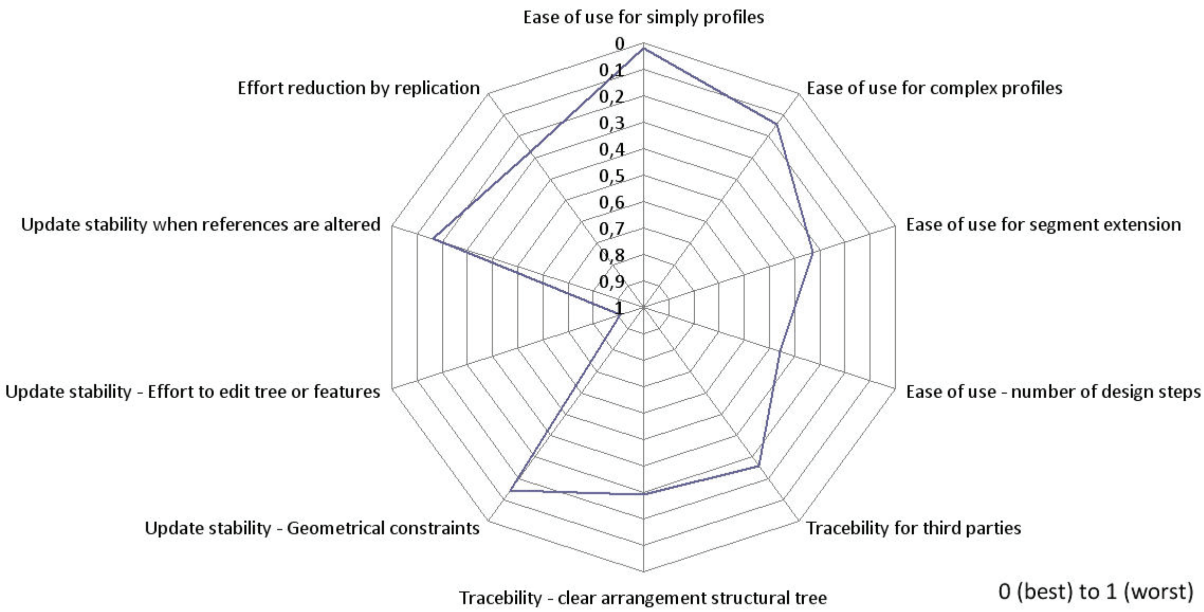


Fig. 7.38 Validation surface band method

In spite of the advantage of update stability the use of this method is not recommended for complex profiles. Compared with the other methods the surface band method is less convincing. The control effort and the confusion grow enormously with the number of segments. A high number of interdependent design steps are necessary which require the control of the directions of curves and surfaces as well as the positions and expansions compared with the concept section after every update. The hierarchical tree gets long and less legible especially for third parties. Actually the sequential design steps are directly legible in the hierarchical tree the interdependencies get more and more inscrutable. Because the design steps and interdependencies are different for the single surface bands a reduction of design effort by replication is only possible in single instances.

With an overall score of 2.98 (1.36 is the score of the best design method) this method is one of the less recommendable of the methods evaluated (see figure 7.38).

7.8 Conclusion to the Chapter

The chapter described a piece of research work regarding the design of profiled zones of automotive body parts and assemblies. Various CAD-models have been designed to improve existing and to test new methods. A detailed description and evaluation shows seven methods for the development of prismatic surface bands necessary for the development of body parts.

The evaluation of the methods for the prismatic design of body surfaces shows that both boundary box methods by far are the best methods evaluated. Especially the boundary box (surface) method scores high with the two important criteria of update stability and traceability. The designer is able to perform the design of several variants without major editing problems. This saves time in the concept phase where lots of variations occur to optimise the designs (see figure 7.39). The necessity to get back to the boundary box (wire frame) method when curved segments are involved is not a problem for the designer. The effort for the many simple design steps can be dramatically reduced by replication of geometrical sets. Another important advantage is that the section is nearly not affected by the extension principle. With other extension principles the general view sometimes completely gets lost.

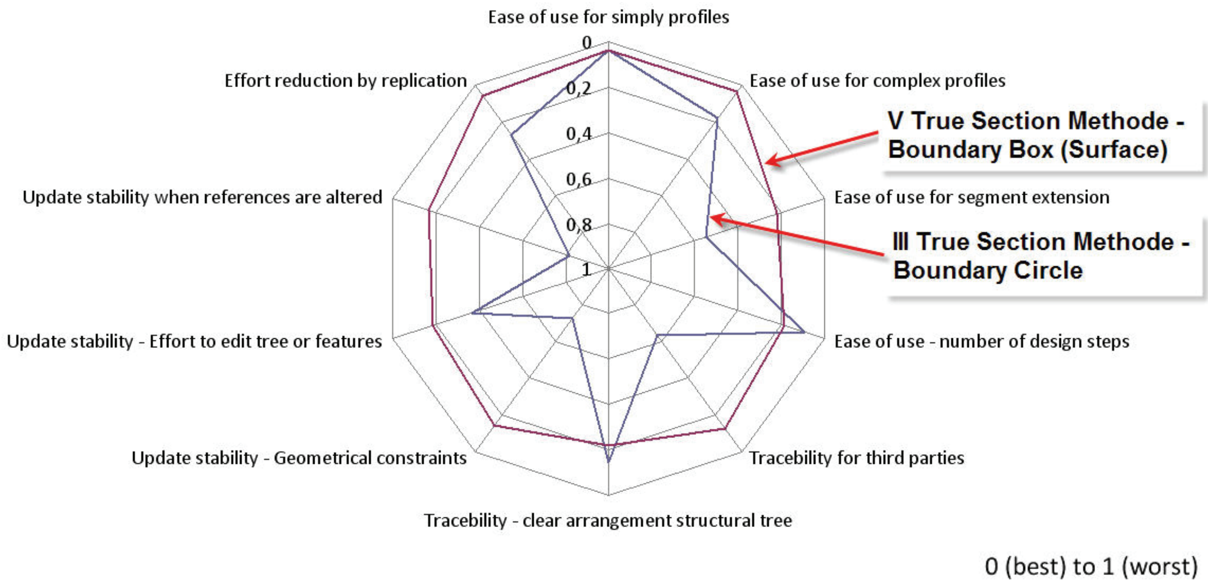


Fig. 7.39 Comparison of best and worst method

For first quick designs in the early concept phase the method boundary box (wire frame) is recommended. This principle can quickly be designed and clearly represents the section and is updating safe.

An alternative for simple as well as complex profiles is the principle “Extrapolation 3D”. The extensions are quick and designed under support of existing points from workbench Sketcher and extrapolation in 3D as long as no curved segments are involved. The small number of design steps keeps a good traceability even for complex sections. Ingenuous designers may use BReps for this principle which will lead to update problems.

As long as the designs are simple the principles “3D only” and “Extension by constraining” are alternatives to boundary box (wire frame). The principle “3D only” is more update safe as long the designer not uses any BReps. The boundary circle principle should not be used at all. The principle often induces failures during editing and with complex sections the section is not legible after the principle is applied.

Chapter 8 Discussion, Conclusion and further Work

The research work for this thesis has been carefully completed after the evaluation of the last three of 25 team projects organised between the author, his students and companies of the automotive industry. The knowledge gained by the author has been completed by several final year projects (>70, duration 5 month) organised between the author, his single students and companies of the automotive industry. In 2008 the author took over a research project from AK 4.6 (collaborative work group of German OEMs) to support the implementation of the CATIA integrated Packaging software programme CAVA in the German automotive industry.

The research of the author in this field of engineering will not end with this thesis. A lot of design and development processes need to be explored and improved. Therefore the author maintains his contacts to the automotive industry.

The automotive industry is very much interested to permanently optimise their processes. A reduction of development time e.g. during concept or tooling phase under support of re-usable PAD models which define part geometries but also control the expected functions is nowadays undisputable. Instead of old 2D controlling processes defined by company regulations and legal requirements, the PAD models deliver 3D geometries to control e.g. the demands of ergonomics, passive safety, manufacturing intents etc. During the period the author worked on this thesis the Volkswagen Group reduced their master PEP from 60 to 48 months.

As there is not very much literature regarding automotive body engineering and the application of PAD in this field the author went to several conferences and became member of the committees for two important conferences which take place every two years. He organised his own conference regarding PAD and invited people from industry to speak at small conferences at the university. Parts of the thesis were sent to specialists in automotive industry for their approval. Two books were published by the author in cooperation with his contact partners from industry.

To understand the motivations and pressure of the automotive industry chapter 1 describes the situation in this industry and the role a body engineer plays in his collaborative work in product development in automotive body engineering. A comparison of old and new design methods shows the chances of a new product development under the support of Parametric Associative Design (PAD). The storage of knowledge in parametric design models is only possible once engineers give up their private specialised knowledge and make it applicable for general use. This leads to the research question of how these processes have to be organised and the explanation of aims, objectives and methodology which have been defined at the early stage of the research work and been slightly readjusted during the work.

To show the differences between pure mechanical design and automotive body engineering basic methods and first examples of automotive body design are shown in chapter 2. These examples point out important features of body design which are prerequisites for a structured Parametric Associative Design (PAD).

The field of PAD is very broad and complex (e.g. wire frame and surfaces versus subdivision modelling, Bezier – surfacing versus NURBS - surfacing, solid modelling versus functional mould design etc.). The large amount of work benches in a PAD software such as CATIA V5 or Siemens PLM NX can be compared with a complex language. To make everybody understand and use the language it is important to compare the advantages and disadvantages of the PAD software workbenches. Joined efforts of the German automotive industry led to a reduced range of functions to make the PAD software understandable and usable for a larger quantity of automotive body engineers. The research work of the author and his students in partial areas of automotive body design supported and supports the decision making process of the automotive industry regarding this subject.

Chapter 3 describes fundamentals of PAD and the author's consolidated findings. Different methodologies to define associated geometries and design features in CAD models of assemblies of an automotive body have emerged during the past few years. This was due to missing opportunities for the satisfactory administration of links between the parts of an assembly in Product Management (PDM) systems. As PAD offers its most advantages in the early phases of development cycle where many variants are approved and several changes appear this led to work with Multi Model Links (MML) method using Reference-to-Instance links in a file based environment at the university as well as in concept engineering teams of the automotive industry.

To keep the structural trees of the PAD neatly arranged, legible and traceable it has proved successful to structure the PAD models according to the development steps. In many cases several development steps can be structured according to production sequences.

In chapter 4 a critical literature analysis of the general systematic approaches to engineering development and design, detailed partial methods of structured 3D CAD modelling as well as priciples of object-oriented design is undertaken. PAD is a type of object oriented design.

The collaborative work group AK 4.6 (CAD/CAM) of the five German automotive companies (AUDI, BMW, Daimler; Porsche, Volkswagen) has elaborated and issued a common “start part structure” in 2009 (AK 4.6, 2009). In the last three team projects the version of the OEM start part tested by AUDI was adjusted for the necessities of the project work in the concept phase to gain an insight of the use of the OEM start part for the structural tree of single parts (see addendum). For sub assemblies and assemblies the consequent use of the IDO (Input-Design-Output)-Principle on all hierarchical levels has proved to be successful.

The design approach and definition of the hierarchical tree requires the consideration of all existing methods and strategies investigated for chapter 4. This approach guarantees the update safe exchange of reference geometry and parameter variations as well as the legibility and the reusability of PAD models of single parts, sub assemblies and assemblies.

To understand the necessities for the application of the PAD methodologies it is essential to get an insight into the various collaborative development works of stylists, designers, CAE engineers and related partners. These are explained in chapter 5. Every unit of the automotive body will be developed with several variants with the objective of improvement towards one optimum model. Every single development step of every variant of every unit needs the design of 3D geometry for evaluation.

Most of the parameters regarding collaborative parametric associative design of assemblies of the automotive body can be investigated and optimised in the file based principle undertaken at the university (HAW). For the distribution of design work in concept development it is useful to organise the design work of assemblies of sheet metal parts and their interior and exterior environment not according to the single parts of the assemblies (bill of material) but according to a zone based approach. Advantages and limits of zone based design have been investigated and presented in detail with the use of examples in chapter 6.

For the development of automotive body parts the use of sections is essential. Chapter 7 investigates the design of profiled zones of automotive body parts and assemblies. Various CAD-models have been designed to improve and to test existing and new methods. A detailed description and evaluation shows seven methods for the development of prismatic surface bands necessary for the development of body parts representative for most other design methods used.

The original research questions were:

1) How does the parametric associative design support the development of body assemblies with distributed tasks?”

This support comes in the form of well structured files of CAD models. Typical examples are described in the addendum (Original files can be found on the DVD).

2) How could a parametric associative CAD model be clearly structured to provide information for linked processes in the form of Knowledge Based Engineering (KBE)?

The investigation has shown that well structured PAD models are capable of containing the knowledge of both the designer and the related processes needed to develop the product.

The original contribution of this research was the application of PAD to the process and design methods of automotive body engineering. The novelty in this approach

and contribution to knowledge was the application of PAD in the process chain of automotive body development where the combination of functions and forms of assemblies often is more complex than in mechanical engineering.
This thesis defined structures where the explicit knowledge of the company and implicit knowledge of the engineers involved in the project, can be captured and transported between various process steps to improve the time and the quality of the output. Rules and standards were defined allowing consistent update safe PAD models which can be re-used in follow up processes and other projects.

The thesis has shown that equal distribution of design work between design engineers in large assemblies can either be subdivided according to the bill of materials or subdivided into zones. The zone based approach was investigated and suitable rules were found specifying the degree of detailing needed in the zones to enable update safe models. Besides the distribution of work the zone based approach has a second advantage: In the past isolated 2D sections (about 180 at Volkswagen) have been taken over from former developments, adjusted and used to start the design of a new automotive body. 2 D sections have been used to control the demands of company regulations and legal requirements. With the zone based approach 3D PAD models from former developments, containing adjustable 2D sections and control sections where necessary, can be taken over and can be adjusted by their parametric associative contents according to new demands. The outcome is not a collection of adjusted sections but an automotive body defined by all important primary surfaces, automatically controlled according to demands of company, legal requirements etc.

To work out and verify the knowledge described in the thesis it was necessary to define lots of PAD models. The addendum delivers a small selection of CAD-model examples which are described in detail.
Two important topics of the collaborative project work which did not play an important role in the projects approached at the university should all the same be observed:
The topic of intellectual property protection in design and development becomes more and more important in the automotive industry. The know-how integrated in parametric associative CAD-models must be protected. The opposite safeguarding of own know-how affects OEMs, suppliers and engineering suppliers. For the handling

of confidential data between OEMs and university a simple way is chosen: The Class-A surfaces (sensitive styling data) are exchanged with existing old Class-A surfaces and internal parameters are manipulated. This way large parametric associative CAD-models can be handled over from OEM to university for further proceedings and project works. The collaboration between OEMs and general contractors or tier 1 suppliers down to engineering suppliers is much more complex and needs individual case studies. In this area highly specialised software companies develop tools for the individual elimination and reintegration of information in CAD-models (Krastel, 2008; Slaby, 2008; Stjepandic, 2008). AUDI works on a process chain adapter, a CA-tool that automatically converts parametric associative CAD-models into tailored models without knowledge. This way it is planned to supply process partners like simulation, external FEM calculation, manufacturing planning, tool making or quality control as early as possible with exactly the current data the partners need (Klem et al, 2008).

OEMs today in the early phase of model validation in product development increasingly use more parametric associative templates to review reoccurring questions. Contrary to the old reactive operation principles these zone based templates are designed for the automatic proactive influencing of the product development to prove and assist the optimising process in terms of process partners. Most of these templates are very complex and need experts for their application. Examples for these templates are worked out for control of legal or package requirements or for safeguarding of intends of ergonomics, passive safety or manufacturing (Brockmeyer, 2010; Potthoff, 2007). The development of templates for early safeguarding of technical demands of process partners as well as the continuous improvement of the findings of the thesis are the essential scopes for future works in team projects in cooperation with automotive industry and other universities.

For 2011 the author has planned to organize three more team projects in the Master programme with AUDI, BMW and Volkswagen. In the Volkswagen project a second university is involved which takes over the role of production planning.

Besides this three team projects a research project is organized for 2011 between AK 4.6 (AUDI, BMW, Daimler, Porsche, Volkswagen) and the university, called : “Investigation of the efficiency of structural design on basis of the OEM start part.”

The main organisational points are:

25 students work in five teams. The distribution of CAD-knowledge is approximately the same in all teams.

Every single student designs the same typical body parts in its concept assembly environment (a sheet metal cross member, the inner trim (sandwich plastic part) of a tale gate, a cross member assembly made of extruded aluminium, a weather strip assembly (extruded profile) for a side door and a moulded (die casting) suspension leg turret).

One team of five student’s works according to OEM start part regulations, another team according to BMW start part regulations and three teams of five students define their own structure in a standard CATPart.

After generation of the five parts, typical modification scenarios, not known at the beginning, have to be implemented. Efforts, experiences and motivations for design, update and repair are carefully registered. The quality of the CAD-models is tested by different tools (e.g. QChecker, Validat) in several stages. The overall result is going to be assessed and published.

CAD Work Benches and Design Guide Lines

List of Content

Content of the DVD (inside rear board cover):

1. PDF-File of dissertation and addendum:

Complete_Dissertation_Gerhard_F_K_Tecklenburg_PhD_2010

2. PDF-Files of MindMaps regarding figures 6.7 and 6.8:

2009-10-07_File-based_Project-Structure_for_Collaboration_
of_Teams_AUDI_and_GFI

2009-10-11_CAD_Model-Structure_for_Collaboration_
of_Teams_AUDI_and_GFI

3. CATIA V5 R19 SP3 models:

2010-02-04_PC_Trapezium_Bead_Rounded_run_out_Final_fillet
2010-02-15_PC_Trapezium_Bead_Straight_run_out_Final_fillet
2010-02-15_PC_Round_Bead_Rounded_run_out_Final_fillet
2010-02-15_PC_Round_Bead_Straight_run_out_Final_fillet
2010-02-04_Test_bench_for_PC_Bead
2010-02-16_UDF_Round_Bead_Rounded_run_out_Final_fillet
2010-02-16_UDF_Round_Bead_Straight_run_out_Final_fillet
2010-02-16_Design_Table_UDF_Round_Bead (MS Excel file related to UDF Files)

2010-04-20_Auslegung_Scharnierachse_vordere_Seitentuer
(Layout_Hinge_axis_Front_door)

2010-01-02_C-Pillar_Test_bench_Prismatic_areas

2010-01-06_POWERCOPY_Development_True_section_and_Surface_bands
Principle_Maurer

List of Tables

List of Figures

Introduction to the Addendum

Throughout the work on this dissertation many CAD models have been developed to test and improve design and structuring methods. Methodologies of body design have been customised and optimised for the PAD process. The contribution of knowledge expected to arise from such studies is the robust integration of PAD into the complex body design scope with its varied relationships along the process chain. Explanations about the re-use and robustness of PAD models give an inside into the success in saving time and money associated with a higher quality of CAD data for the development process expected from the investigations. Three additional examples give a small overview about the methodologies of automotive body design investigated:

- Appendix 1: The templates “Prismatic design of trapezium and round swages” exemplify how the design work for the high number of embossments on structural body parts can be easily repeated to save development time. These templates are used in the part-phase of concept development to apply stiffening swages to sheet metal parts (compare with chapter 6).
- Appendix 2: The concept development “Layout hinge axis front door“shows an example for the layout work of a typical area of an automotive closure. This CAD-model is used for the layout of hinge axis in the zone-phase of concept development (compare with chapter 6).
- Appendix 3: The template “From concept-section to true section and surface bands” shows an automated procedure which saves a lot of time for the development of typical surfaces of automotive body parts. Chapter 7 describes the methodologies of sections and derivation of surfaces. This template is automatically developing surface bands from true most concept sections and can be used in zone phase of concept development (compare with chapter 6).
- Appendix 4: The execution of several team projects intensified the comprehension that a collaborative design work is only possible with clearly defined design guide lines. Guidelines applied in the projects presented in chapter 6 are presented in full detail this Appendix.

Appendix 1 Design of Prismatic Swages

A.1.1 Introduction

Swages (beads) are hollow shaped embossments mainly in plane or flat curved sheet metal parts (see figure A.1.1) which enhance the stiffness and allow weight reduction by the use of thinner metal sheets. Sometimes styling and enhancement of stiffness is combined on visible body parts (e.g. rear wall, cabin, pick-up or covering sheet, bumper, SUV). In deep draw process swages are used to reduce local wrinkle formations.

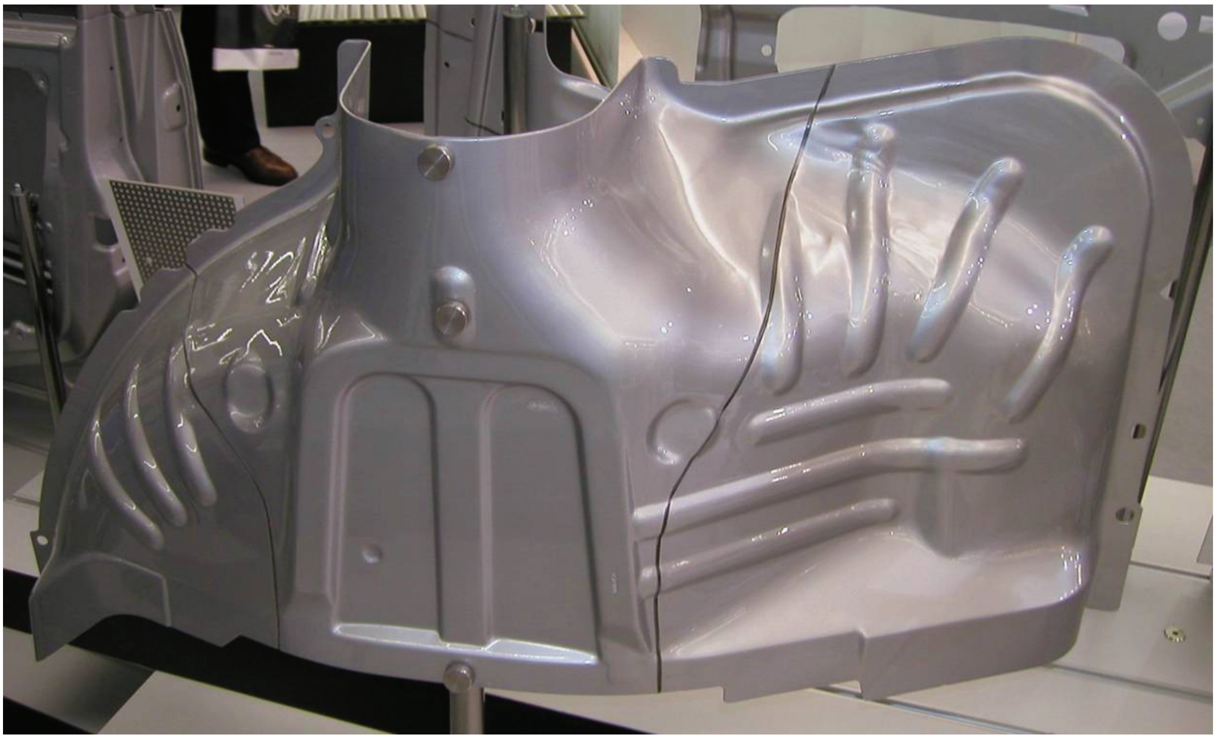


Fig. A.1.1 Wheel arch FORD Mondeo

From the geometrical or design point of view swages are only prismatic (of constant cross section) as long as their spine and guide curves are identical. The cross section does not change along the route of a swage. For swages designed according to manufacturing technology the spine curve is positioned on the tooling plane and the guide curve on the body surface the swage is applied to. These swages are not prismatic. The cross section changes its shape along the route of the swage. The design of prismatic round swages requires a special attention. The small angle at the end tangents on the border between swage and reference surface may lead to local undercuts (see figure 7.4 in chapter 7).

The profile shape and the dimensions of the cross section are depended on the sheet thickness, the number and position of necessary swages. These are investigated by simulation. In this chapter the design of trapezium (see figure A.1.2) and round swages according to Volkswagen standard is described. The high number of swages on structural body parts requires design methods which can be easily repeated (e.g. templates like *Power copies* and *User Features*). As the route of a swage can be straight or curved the templates are defined to fulfil both requirements.

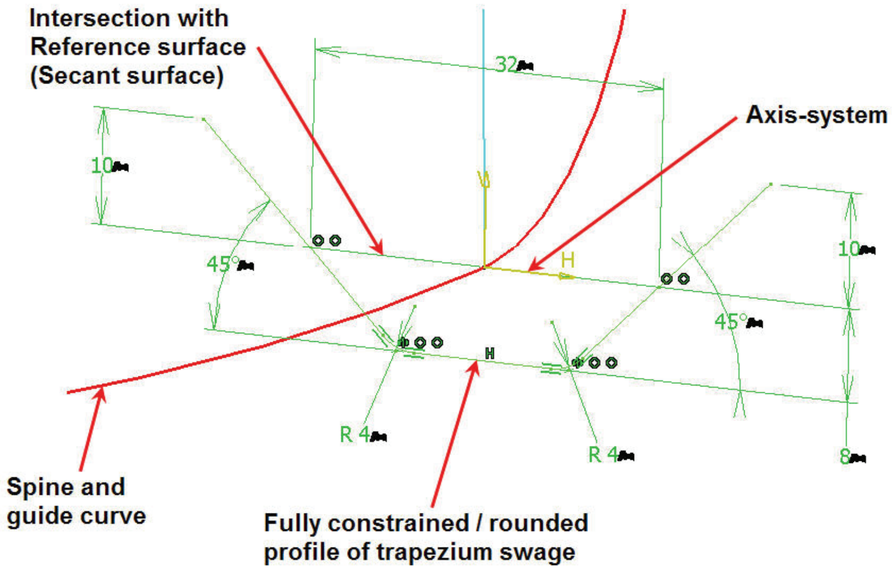


Fig. A.1.2 True section (compound section) of a prismatic trapezium swage

A.1.2 Design of Templates

The structural tree of a template looks different than of a normal CAD-model. On the first hierarchical level there are two files, one for INPUT geometries and one for the design to be repeated (see figure A.1.3: Trapezium swage with rounded run out). The files for DESIGN and OUTPUT are defined on the second level below the file for the design to be repeated.

The INPUT geometry always must be inserted isolated. It is important not to generate links or update loops with the geometry the template is applied to, especially when the template is developed on basis of the same CAD-model. In *CATIA* there are several possibilities to automate repeatable design applications. In this case by way of example *Power copies* and *User Features* are used to design swages. While the structural tree of the repeated geometry of a *Power copy* is added to the structural tree of the CAD model the template is applied to can completely be edited. The structural tree of a *User Feature* is not available in the CAD model after the template has been inserted and cannot be edited. Therefore *User Features* are often used for designs according to standards where parameters are not allowed to be changed (e.g. VW-Standard 01080). As standards e.g. for material or semi finished products such as sheet metal used in automotive body construction are often not up to date it is advisable to develop a *User Feature* for standard applications as well as a *Power copy* for certain variations.

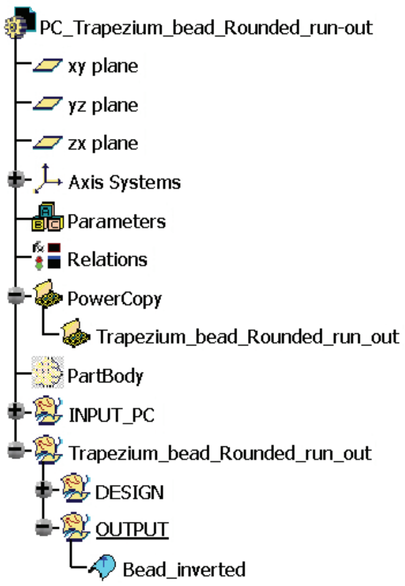


Fig. A.1.3 Hierarchical tree of a template. INPUT, DESIGN and OUTPUT files are positioned on different hierarchical levels.

As the Volkswagen-standard allows several variations for the design of trapezium swages (Form B, see figure A.1.4) two *Power copies* are designed for these swages which show both shapes of run-out mentioned in the standard, round run out shape and straight run out shape. The *Power copies* are completed with a table of dimensions and tolerances of the Volkswagen- standard. According to this table the *Power copies* can be edited manually if necessary. For round swages (Form A, see figure A.1.4) *User Features* are defined which automatically apply the dimensions of the standard according to the sheet metal thickness of the part the template is applied to.

For the application of round swages in sheet metal parts which are not according to the standard two more *Power copies* are designed.

Reaction.1 : Information

Dimensions Form A (according to Volkswagen Standard 01080)

h1)	2 ±0.5	3 ±0.5	4 ±1	5 ±1	6 ±1.5	8 ±2	10 ±2
r1	2.5 +1	4	5 +2	6	8	10 +3	12
r2 2)	1.6	2	2.5	2.5	3.2	4	4
t	0.6 to 1	0.8 to1.2	1 to1.5	1.2 to2	1.5 to2	1.5 to2	2 to2.5

Dimensions Form B (according to Volkswagen Standard 01080)

h1 1)	2 ±0.5	3 ±0.5	4 ±1	5 ±1	6 ±1.5	8 ±1.5
a	12 16 20 ±1 ±1.5	16 20 ±1 ±1.5	16 20 25 32 ±1 ±1.5 ±2	16 20 25 32 ±1 ±1.5 ±2	20 25 32 ±1.5 ±2	25 32 ±1.5 ±2
r2,r3 r2 2), r3	1.6 +2	2 +2	2.5 +3	2.5 +3	3.2 +3	4 +3
α	45°	45°	45° 60° ± 5°	45° 60°	45° 60°	45° 60°
t	0.6 to 1.2	0.6 to 1.2	0.6 to 1.2	0.8 to 2.0	0.8 to 2.5	0.8 to 2.5

1) The tolerances apply to individual beads. If there are several beads next to one another, the tolerances increase by 50%.
2)The radius r2 is dependent on the degree of deformation and the material; as a result it cannot always be adhered to.

OK

Fig. A.1.4 Table of dimensions and tolerances of swage cross sections according to Volkswagen standard

A.1.3 Design of *Power Copies* for trapezium and round Swages[1]

Firstly isolated geometries (see figure A.1.5) must be provided in the INPUT file of the template. For the reference surface any type of surface can be designed or taken over from another CAD-model. A tooling plane is added to the reference surface to

[1] Please see the following CAD-models:
2010-02-04_PC_Trapezium_Bead_Rounded_run_out_Final_fillet
2010-02-15_PC_Trapezium_Bead_Straight_run_out_Final_fillet
2010-02-15_PC_Round_Bead_Rounded_run_out_Final_fillet
2010-02-15_PC_Round_Bead_Straight_run_out_Final_fillet

support the design of the next references. The plane will be deleted as soon as the references are isolated. The starting point and route of the swage have to be designed either on the reference surface or on the tooling plane of the reference surface. A straight line perpendicular to the tooling plane defines the direction of swaging operation. This straight line is positioned on the opposite side of the planned embossment.

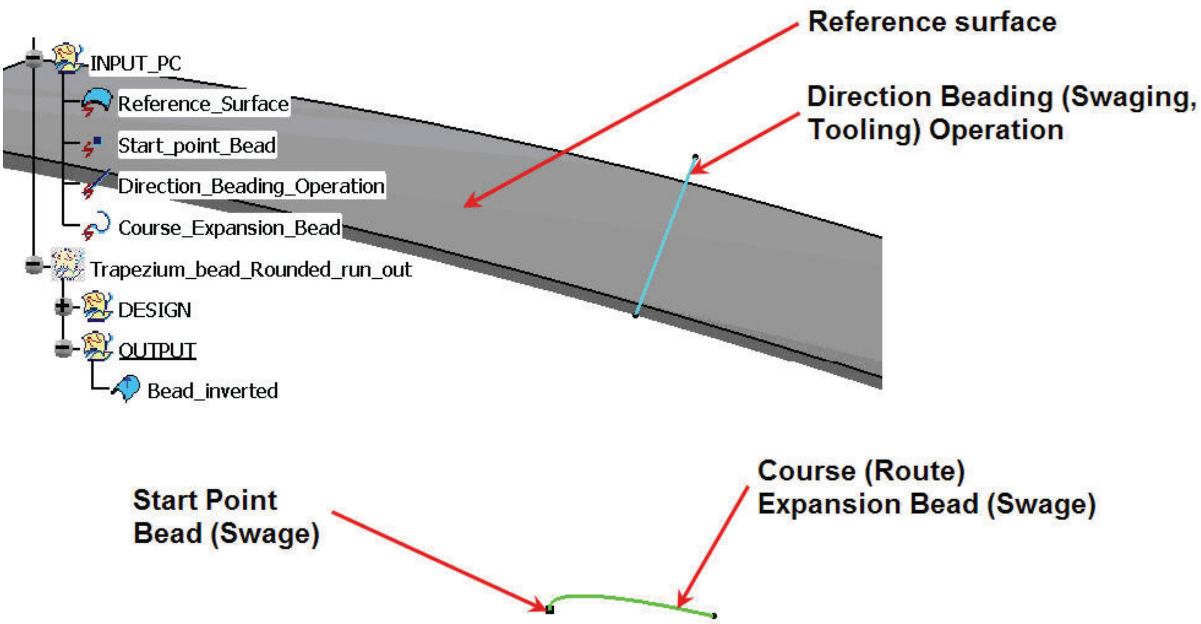


Fig. A.1.5 INPUT Power copy with isolated geometry of reference surface, start point, direction swage operation and route expansion swage

The explicit parameters of the trapezium swage are defined on basis of Volkswagen standard (see figure A.1.6). The dimensions shown are valid for a sheet metal thickness of 2.5 mm. For the slope angle dimensions of 45° or 60° are specified. For the run out radius a value range from R25 to R40 is predetermined. For the straight run out the standard defines a slope angle of 12° and a fillet of R60. To make sure that the surface of the swage can be error free when filleted with the reference surface, a parameter for the extension of the segments of the cross section is defined. The explicit parameters for the round swage are defined in a similar way.

For the integration of the table with the dimensions and tolerances of the Volkswagen-standard the KBE feature *Reaction* is chosen. Therefore first of all an explicit parameter of type *String* (switch) has to be defined to provide the two values

“Form A“and “Form B”. For this parameter a *Reaction* will be defined which shows a message as soon as the value is exchanged from “Form A“to “Form B” or contrary. The message is the table of dimensions and tolerances copied from Volkswagen standard (see figure A.1.4).

Fig. A.1.6 Explicit parameters of trapezium swage with round run-out

The explicit parameters defined will be applied by *CATIA* to the standard parameter set. As the parameters have to be placed in the DESIGN file of the *Power copy* an additional parameter set is defined, cut and pasted into the DESIGN file. The explicit parameters defined in the standard parameter set before are reordered into the new parameter set.

After the start point and the route of expansion are projected onto the reference surface and the end point of the projected centre curve is designed, two local axis systems are defined in start and end point of the centre curve (see figure A.1.7). Therefore in both points normal planes are designed perpendicular to the centre curve and intersected with the reference surface. The X-axis (width of swage) will be defined on the normal plane as a tangent to the intersection curve. The Y-axis (depth of swage) will be designed on the normal plane perpendicular to the X-axis. The Y-axis has to be positioned on the same side than the predefined direction of swaging operation. The Z-axis is a tangent to the centre curve.

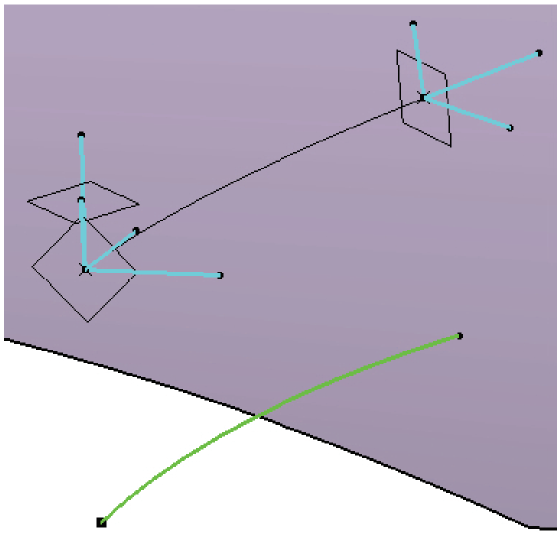


Fig. A.1.7 Local axis system at start and end point of swage centre curve

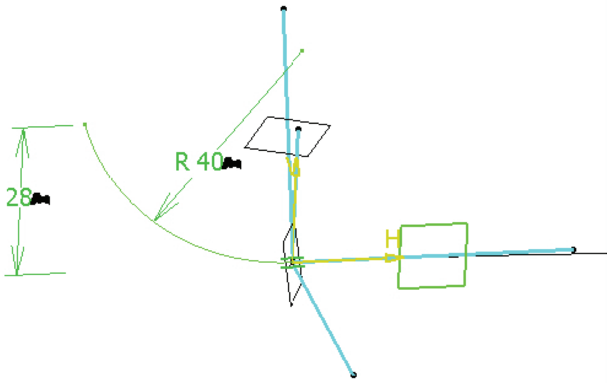


Fig. A.1.8 Round run out of swage

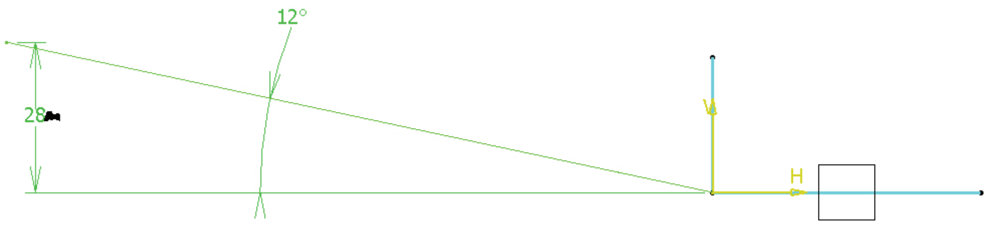


Fig. A.1.9 Straight run out of swage

Under support of the Y- and Z-axes of the local axis systems two planes are defined to design sketches for the round or straight run out (see figures A.1.8, A.1.9). The spine and guide curve of the swage now consists of three segments, the projected route of expansion and the plane run-out shapes on both sides. As it is necessary to design a curvature continuous spine and guide curve several design steps are required. At first the projected route of expansion will be extrapolated on both ends curvature continuous. Under support of the direction of swaging operation the curvature continuous middle surface for the swage is extruded. The two run out shapes will be projected onto the middle surface and connected or filleted with the centre curve. In a *Curve Smooth* - operation a curvature continuous spine and guide curve is designed on basis of the prepared tangent continuous curve.

On a normal plane in the start point of the centre curve a true section (compound section) of the swage cross section is designed (see figure A.1.2). For the round swage the cross section is a closed circle. The swage surface will be defined under support of the cross section along the curvature continuous spine and guide curve.

There are two possibilities for the final result of the swage design. Either the swage surface is the final result positioned in the OUTPUT-file of the structural tree or the fillet between the swage surface and the reference surface is the final result. The last version should only be used if a single swage is to be implemented in a sheet metal part. To avoid conflicts with complex surfaces in all other cases (e.g. several swages in one part, crossing swages) the swage surface should be the final result of the swage template and the fillet work should be done manually. To avoid weakening of the sheet metal structure swages should preferably not cross each other.

After a concluding in-depth investigation of the consistency of the three sketches involved the *Power copy* is defined. The combination of two different cross sections and two different run out shapes leads to *Power copies* for four different swages (see figure A.1.10). Spine and guide curve as well as cross section are kept visible. This allows to easily identifying one of several swages in the sheet metal part. When one of the two generating elements is selected the swage will be highlighted in the structural tree. When the name of the swage is selected in the structural tree the generating elements of the associated swage will be highlighted. After the *Power copy* is defined its application is tested in another CAD model[2].

[2] Please see the following CAD-model:
2010_02-04_Test_bench_for_PC_Bead

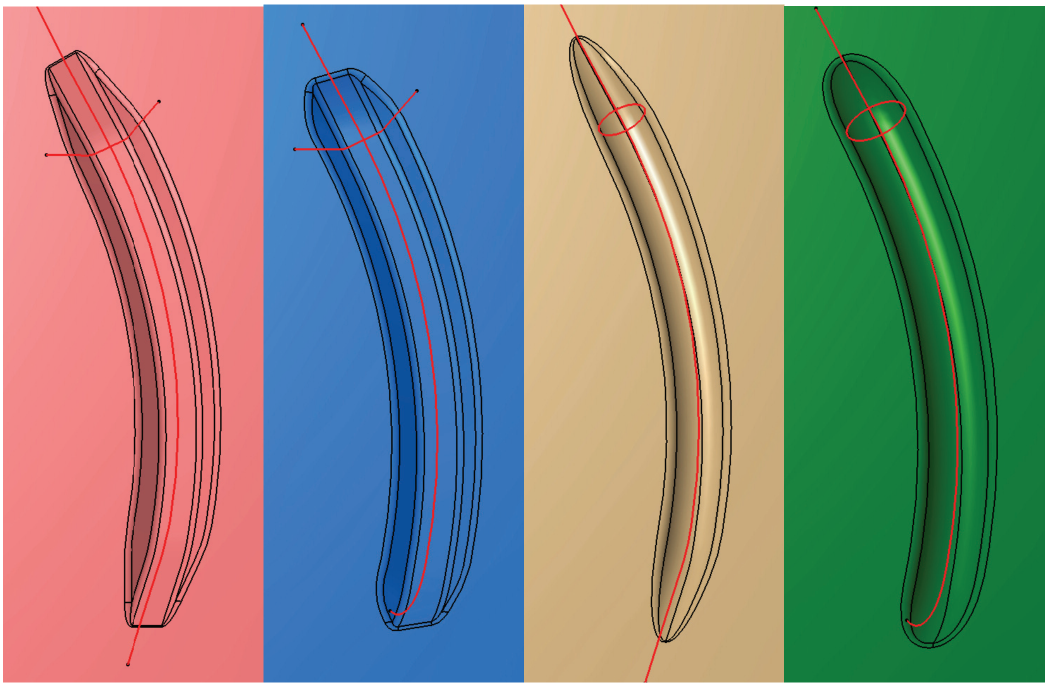


Fig. A.1.10 Trapezium- and round swages with straight and round run out

A.1.4 Design of *User Features* for round Swages[3]

Fig. A.1.11 INPUT_UDF with control parameter for screening the thickness of the sheet metal part the template is applied to

The finished *Power copies* for the round swages will be changed and completed to configure *User Features* out of them. Therefore at first the *Power copy* and the *Reaction* are deleted from the CAD-model. The INPUT-file is be completed with a

[3] Please see the following CAD-models and MSExcel-file:
2010-02-16_UDF_Round_Bead_Rounded_run_out_Final_fillet
2010-02-16_UDF_Round_Bead_Straight_run_out_Final_fillet
2010-02-16_Design_Table_UDF_Round_Bead

control parameter for the metal sheet thickness to enable the *User Features* to ask for the thickness of the metal sheet the *User Feature* is applied to (see figure A.1.11).

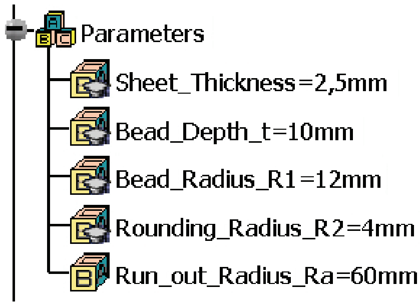
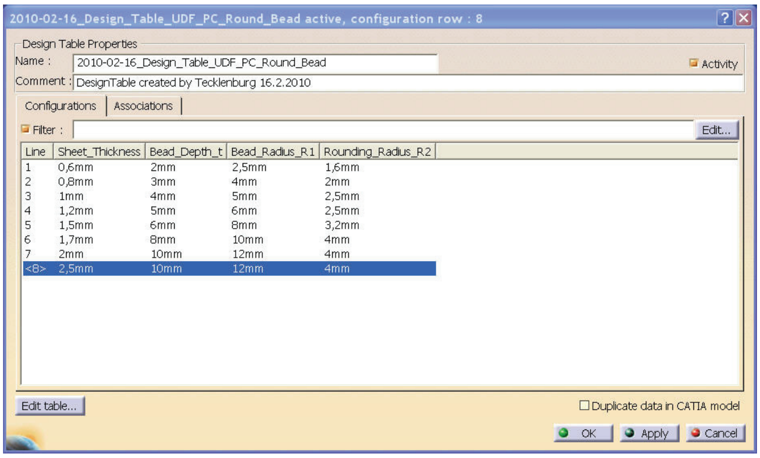


Fig. A.1.12 Explicit parameters of round swages completed by a parameter for sheet thickness

Based on the explicit parameters of the round swage (see figure A.1.12) a design table is defined (see figure A.1.13). According to the Volkswagen-standard eight parameter families (configurations) are organised on basis of the eight metal sheet thicknesses of the standard. The explicit parameters and the design table are linked to each other. As soon as a new parameter family from the design table is chosen the list of explicit parameters in the structural tree will be altered and the design geometry will be updated.



Line	Sheet_Thickness	Bead_Depth_t	Bead_Radius_R1	Rounding_Radius_R2
1	0,6mm	2mm	2,5mm	1,6mm
2	0,8mm	3mm	4mm	2mm
3	1mm	4mm	5mm	2,5mm
4	1,2mm	5mm	6mm	2,5mm
5	1,5mm	6mm	8mm	3,2mm
6	1,7mm	8mm	10mm	4mm
7	2mm	10mm	12mm	4mm
<8>	2,5mm	10mm	12mm	4mm

Fig. A.1.13 Design table for control of parameter families depending on sheet thickness

Under support of the rule editor a rule is defined in *CAT Script* (see figure A.1.14) to link the control parameter for the sheet thickness of the INPUT-file to the design table. When a standard parameter is selected for the control parameter the rule automatically picks out the parameter family associated with, the explicit parameters in structural tree are synchronised and the design geometry is updated. If the chosen parameter does not suit to the standard the predefined error message will appear on the screen and the design sequence of the swage will be deactivated.

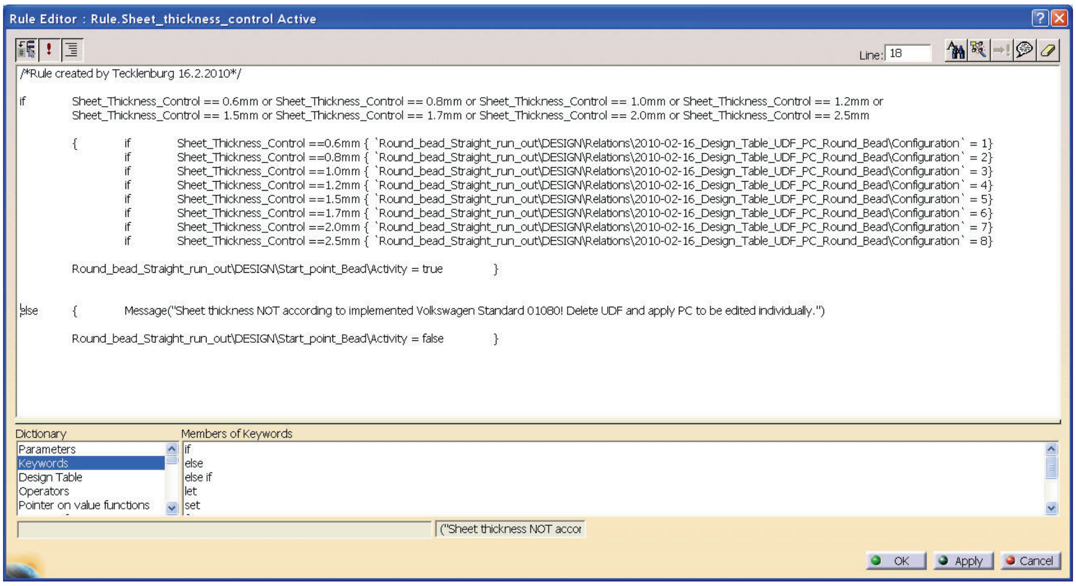


Fig. A.1.14 Rule editor with rule for control of swage cross section according to thickness of sheet metal part the template is applied to

After a checkout of the functions implemented the *User Feature* is defined and tested in another CAD-model (see figure A.1.15). The exemplary insertions show a *Power copy* on the left hand and a *User Feature* on the right hand sight. The structural tree of the *Power copy* inserted is fully accessible and editable. Spine and guide curve are visible and after insertion of several *Power copies* these elements support the allocation of geometry and structural tree of every single *Power copy*. The *User Feature* is shown in the structural tree only as a result with its Boolean parameter “Activity”. The structural tree and the spine and guide curve of the *User Feature* are not visible and editable. The identification of the *User Feature* can only be carried out from structural tree of the CAD-model and not from its designed geometry.

A.1.5 Summary

In this chapter the design of templates for the replication of swages is described. The combination of two different cross sections (trapezium and circle) and two different run-out shapes (round and straight) leads to four basic types (see figure A.1.10). All four are fully editable *Power copies*. The two templates for round swages are upgraded to *User Features* which under support of a rule and a design table automatically change the dimensions of the cross section according to the sheet thickness of the sheet metal part the template is applied to.

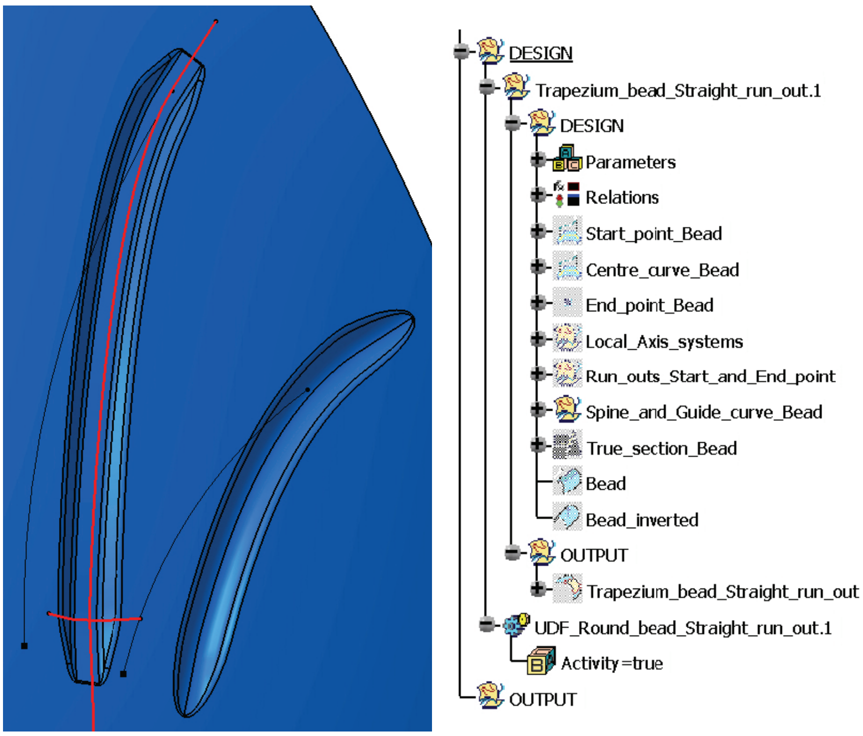


Fig. A.1.15 Work bench with *Power copy* and *User Feature*

To avoid conflicts with complex surface geometries when more than one swage template is inserted the templates should only be applied as swage surfaces and filleted manually. From the four template variants described above another set of templates is provided which deliver as final result the swage surface only. The functions and shapes of all eight template variants could be combined in one template and its single functions e.g. could be switched “true” (on) and “false” (off) by their Boolean operator “Activity”. However for the development of templates it is necessary to keep the complexity as small as possible to avoid conflicts during application especially when a *User Feature* is used which cannot be edited. Therefore it is better to provide a set of templates with low grades of complexity to cope any kind of application.

Appendix 2 Layout Hinge Axis Front Door[4]

[4]Please see the following CATIA-model:
2010-04-20_Auslegung_Scharnierachse_vordere_Seitentuer (Laout_Hinge_axis_Front_door)_Tec

A.2.1 Introduction

During design and early concept phase it is necessary to check the position of the door gap or to define a bounding curve for the position of the door gap in the hinge area. The distance between the vertical hinge axis and the widest part of the sidewall and the relative position hinge axis are taken over from a previous carline (see figure A.2.1).

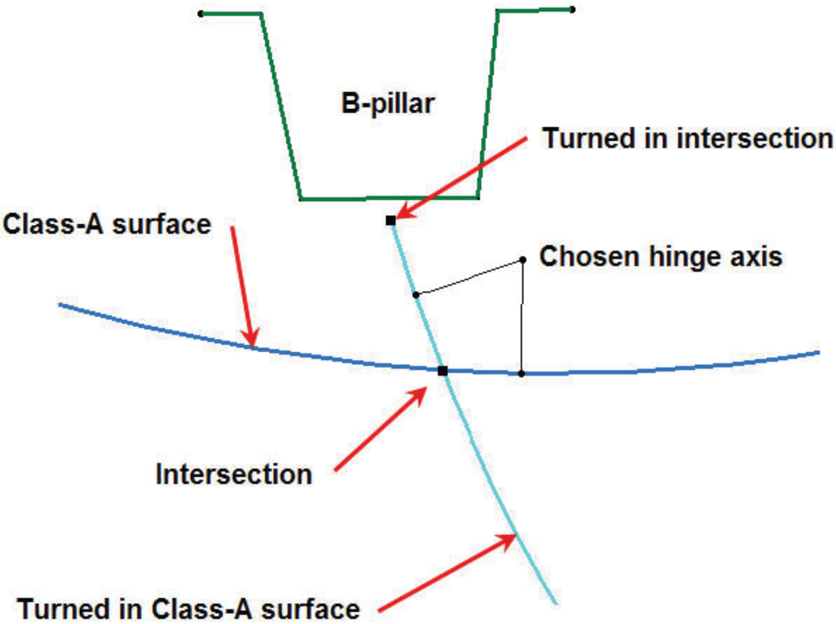


Fig. A.2.1 Layout hinge axis design phase

When during the concept phase more and more constraints and meeting parts such as pillar, hinges, door gap etc. are defined it is necessary to exactly define the hinge axis under consideration of meeting parts, explicit parameters and tolerances. Therefore in most cases an envelope of bounding surfaces for the positioning of the hinge axis is designed. Alternatively a positioning envelope for the door gap can be designed under consideration of a designed hinge axis, pillar and tolerances when the Class-A surface has been drastically altered (see figure A.2.2).

In this appendix the parametric associative approach of the mostly used method for the design of a positioning envelope of the hinge axis is described (Brockmeyer, 2005 and Fischer, 2006:2).

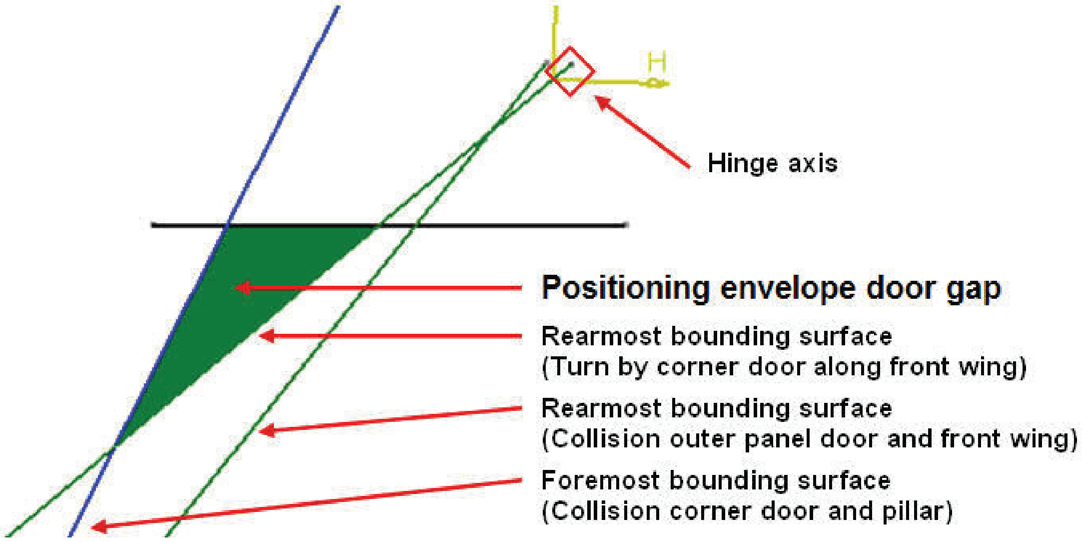


Fig. A.2.2 Positioning envelope door gap (Fischer, 2006:2)

Side doors of passenger cars generally have a turn in arrangement for the hinge to door position. This means that parts of the door which are situated outside or in front of the hinge axis may turn into the sidewall while the door is opened.

To allow the adjustability of the hinges in X-, Y- and Z-direction most hinges are fixed on the body in white side on a planar surface parallel to the ZX-plane and on the door side on a planar surface parallel to YZ-plane. When the door is opened the door corner may collide with the hinges in this arrangement.

Hinge axes of side doors are often slightly inclined around X- and Y-axes. With angles of inclination up to 1.5° of the upper axis point to the rear and to the inside the rearmost lower corner of the door will be lifted up when the door is opened and the dead weight of the door will support the closing of the door.

With the simple design method mentioned in introduction the early Class-A surface is turned around the hinge axis (see figure A.2.1). The intersection curve of the turned and unturned Class-A surfaces is the rearmost bounding curve for the positioning of the door corner in the area of hinge axis. When a door corner is chosen behind this bounding curve the outer panel of a front door will collide with the rear corner of the front wing. Therefore it is necessary to choose a door gap on or in front of the bounding curve. After the door is opened the front corner of the opened door defines a bounding curve for the positioning of the outer surface of the pillar.

For the design method discussed the following references (meeting parts, constraints) are already defined: Class-A surface with door corner, outer surface of hinge pillar, hinges and the shape of the door flange (crash profile or poor hemming profile).

On basis of these references the following bounding surfaces are designed:

- Rearmost bounding surface collision door corner with outer surface pillar
- Rearmost bounding surfaces collision door corner with hinges
- Foremost bounding surface collision door outer panel with front wing
- Foremost bounding surface turn by corner door at front wing
- Outmost bounding surface hinge axis to Class-A surface

A.2.2 References

The references are taken over from former theses (Brockmeyer, 2005 and Fischer, 2006:2). The data of four different passenger cars are available in the CAD-model. The references of the 5 series BMW limousine (E60, 2003–2010) can completely be replaced by the references of one of the other passenger cars or by own references defined for a new project.

The defined references of the four cars are:

- Class-A surface in the area of the front door,
- Front corner of the door,
- Outer surface of the A-pillar.

The complete outer Class-A surfaces of a new project must be reduced to the area between waist line and sill in height and between front wheel and B-pillar in length.

For three of the four reference cars the controlling references of the series hinge axis is available. The hinges are designed in this CAD-model. The designed hinges may be replaced by Carry-Over-Parts (COP).

A.2.3 Design of the Positioning Envelope for the Hinge Axis

The positioning envelope for the hinge axis (see figure A.2.3) is defined by a foremost, a rearmost and an outmost bounding surface. In addition the positioning envelope is restricted by the rearmost bounding surfaces regarding the collision surfaces of the hinges.

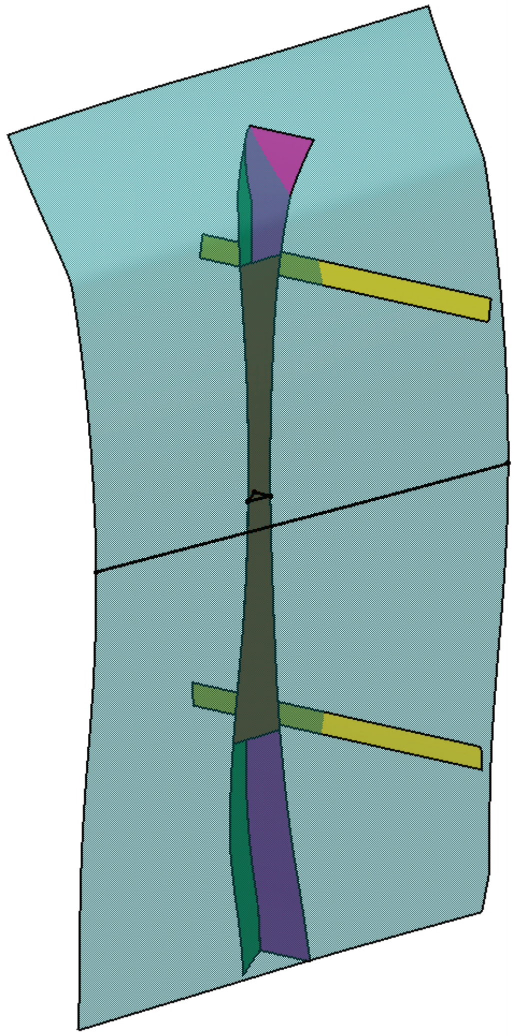


Fig. A.2.3 Positioning envelope hinge axis and Class-A surface front door

There are two foremost bounding surfaces. If the door gap of a front door is relatively far back the outer door panel will collide with the rear corner of the front wing. In this case, from the two foremost bounding surfaces the surface “collision door outer panel with front wing” is chosen. In section A.2.3.5 of this chapter it is described how the selection between the two foremost bounding surfaces is worked out. The bounding surfaces are trimmed to each other to build the positioning envelope.

A.2.3.1 Rearmost Bounding Surface Collision Door Corner with outer Surface Pillar

The hinge axis of a door must be positioned in a way that the theoretical front corner of the door under consideration with defined clearances never collides with the hinges or the outer pillar surface. Firstly the collision with the outer pillar surface is examined.

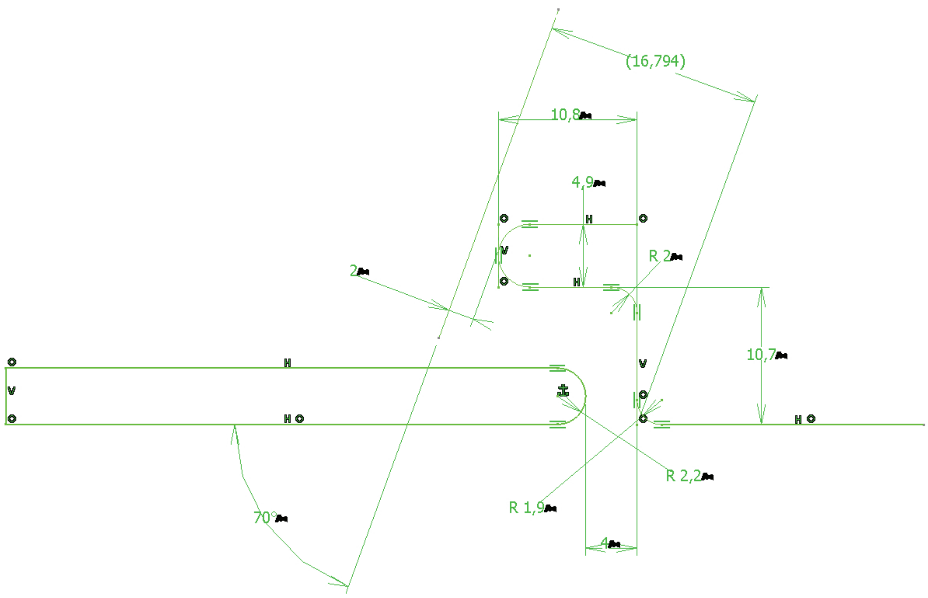


Fig. A.2.4 Reference section corner door with crash profile – Under consideration of door opening angle 70° and clearance (2mm) the minimum distance between theoretical door corner and outer pillar surface is 16.794 mm.

Most side doors with turn in arrangement show a crash profile along the lower front corner of the door (see figure A.2.4). This shape of the door flange prevents the rear corner of the front wing at a front side door or the rear corner of a front door at a rear side door, to slip inside of the door flange in case of an accident. The door flange always has to be inside the opposite flange, so in case of an accident allows the door flange to turn into the sidewall while the door is opened. As there are still cars on the market without crash profile (see figure A.2.5) the CAD-model allows the user to choose between the two versions. In the CAD-model an explicit parameter with switch function is defined allowing switching from one to the other version. A comparison of figure A.2.4 and A.2.5 shows for an identical door opening angle that the outer pillar surface must lie far more inside as soon as the crash profile is provided. If the outer pillar surface is already designed the hinge axis for a side door with crash profile must lie far more outside.

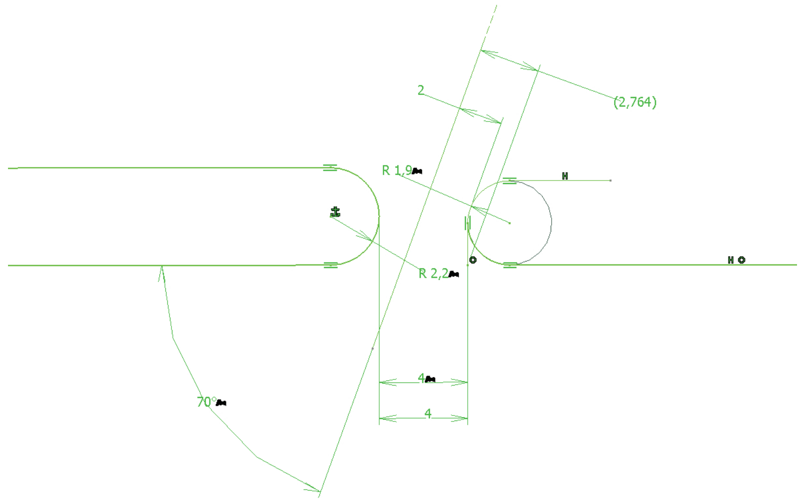


Fig. A.2.5 Reference section corner door without crash profile – Under consideration of door opening angle 70°and clearance (2mm) the minimum distance between theoretical door corner and outer pillar surface is 2.764 mm.

For the placement of the hinge axis between outer pillar surface and Class-A surface there are two theoretical borderline positions (see figure A.2.6):

In horizontal sections the hinge axis can either be placed on the outer pillar surface or on the Class-A surface. A swept auxiliary surface is designed along the front corner of the door under the predefined opening angle. The swept surface intersects with the outer pillar surface along a bounding curve which describes the innermost position of the hinge axis (hinge axis 1). If the front corner of the door is turned in horizontal sections around points of this curve, the front corner of the door always will collide with the outer pillar surface when the defined door opening angle is achieved.

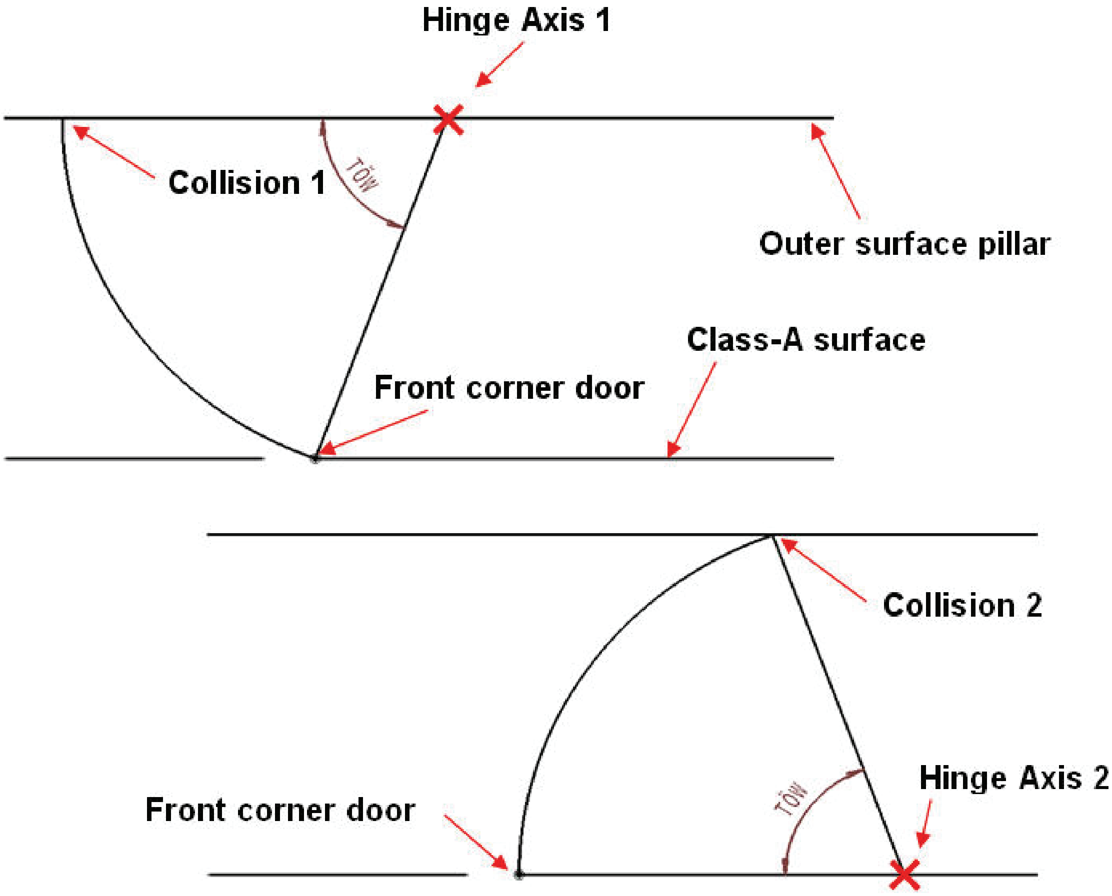


Fig. A.2.6 Theoretical investigations for borderline positions of hinge axis for collision of front corner door with outer surface pillar (Fischer, 2006:2)

To define the second bounding curve for the outmost position of the hinge axis (hinge axis 2) the distance between front corner door and first bounding curve can be measured and turned onto the Class-A surface in many horizontal sections. It is much easier to design a second swept auxiliary surface on basis of the first bounding curve which intersects with the Class-A surface along the outmost bounding curve for the hinge axis (hinge axis 2).

From a theoretical model with vertical planar outer pillar surface and parallel Class-A surface (see figure A.2.7) the complementary angle for the design of the second swept auxiliary surface can be determined. In several horizontal sections this complementary angle can be proven in the CAD-model with its planar outer pillar surface and free-formed Class-A surface. At a door opening angle of 70° the complementary angle is 55° ((180° - door opening angle) / 2) used to design the second swept auxiliary surface and the second borderline position for the hinge axis.

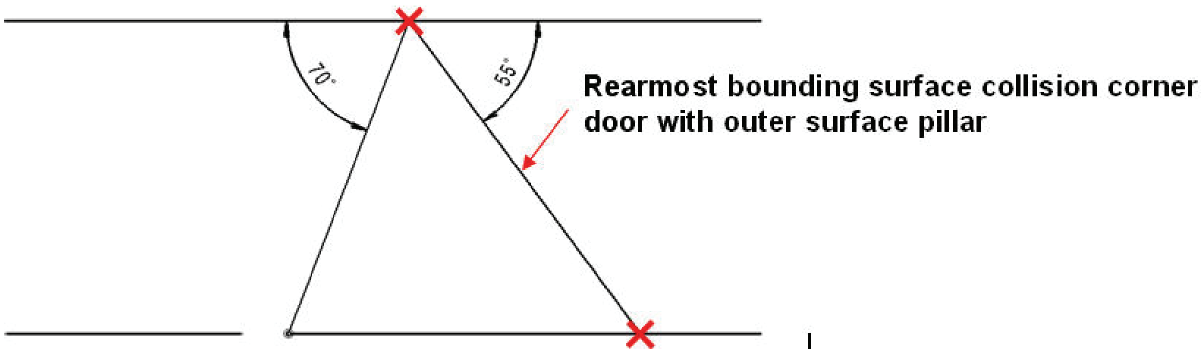


Fig. A.2.7 Theoretical investigation for the angle of the rearmost bounding surface with planar outer surface pillar and parallel planar Class-A surface (Fischer, 2006:2)

In the CAD-model a reference section is defined (see figure A.2.8) which controls the complementary angle in relation to the door opening angle. The second swept surface which connects the two borderline positions is the rearmost bounding surface for the collision of front corner door with the outer pillar surface. All hinge axes positioned along this surface will lead to a collision of front corner door with the outer pillar surface.

Firstly all according to reference section of variant 1 (figure A.2.4 - door flange with crash profile) or reference section for variant 2 (figure A.2.5 - door flange without crash profile) an offset surface of the outer pillar surface is designed to be used for the follow up collision investigation described above. The reference of front corner door is extrapolated beyond the waist line of the Class-A surface to ensure that the extrapolated curve always can be trimmed by the offset surface of the outer pillar surface (see figure A.2.9). To make sure that Class-A surfaces can be used in the CAD-model which does not provide horizontal sections parallel to the outer pillar surface two more auxiliary surfaces have to be designed. The trimmed front corner door is projected in Y-direction onto the offset of the outer pillar surface. Between the trimmed front corner door and its projection the first auxiliary surface is defined. Perpendicular to the first auxiliary surface, a surface band is defined along the trimmed door opening line which runs in horizontal sections parallel to the ZX-plane.

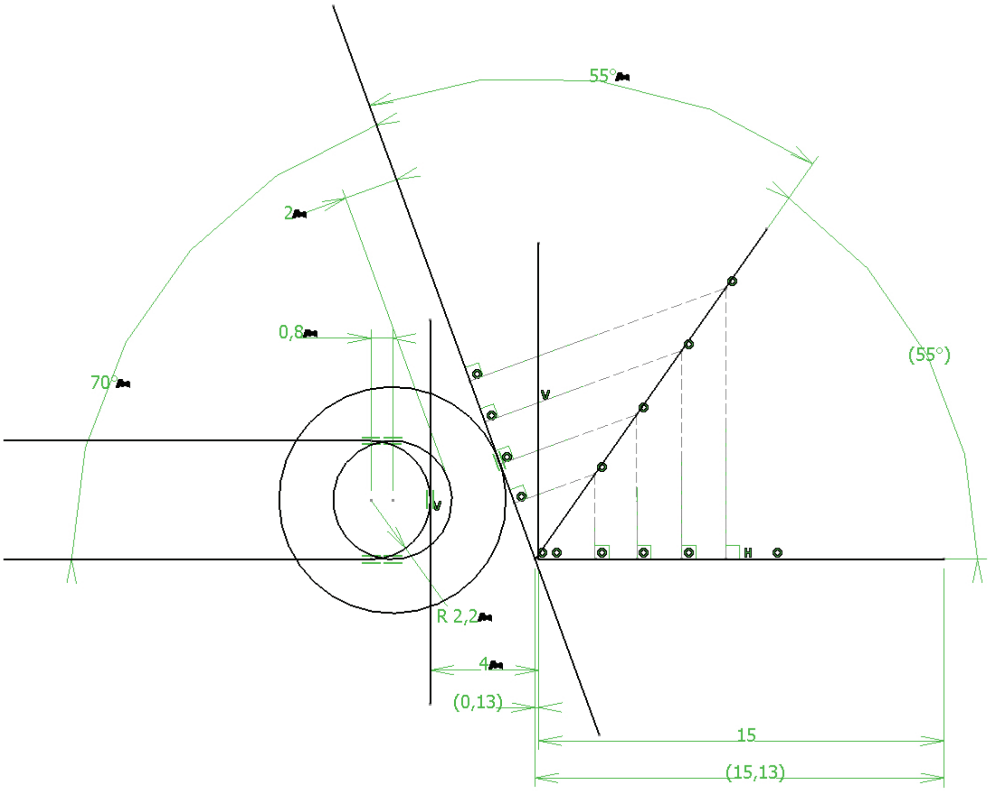


Fig. A.2.8 Reference section to control complementary angles and distances

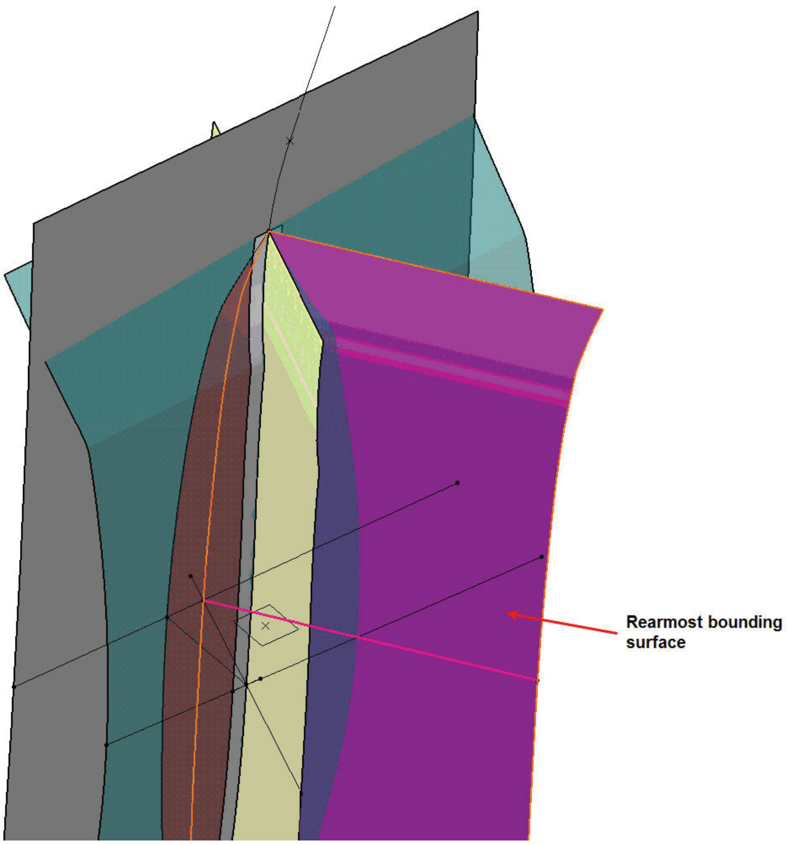


Fig. A.2.9 Design of rearmost bounding surface

From the surface band the first swept surface is defined along the trimmed door opening line which intersects with the offset of the outer pillar surface. The intersection curve is the first borderline position for the hinge axis. From the ZX-plane another swept surface is designed along the intersection curve under the complementary angle (for 70° the complementary angle is 55°). The swept surface is the rear most bounding surface collision corner door with outer surface pillar. All hinge axes positioned behind this bounding surface will lead to a collision between door and outer surface pillar.

A.2.3.2 Rearmost Bounding Surfaces Collision Door Corner with Hinges

The hinge axis of a door must be positioned in a way that the theoretical front corner of the door under consideration of defined clearances never collides with the hinges or the outer pillar surface. Often the hinges are mounted to the outer surface of the pillar. From this it follows that the door flange first collides with the hinges or the mounting parts when the door is opened.

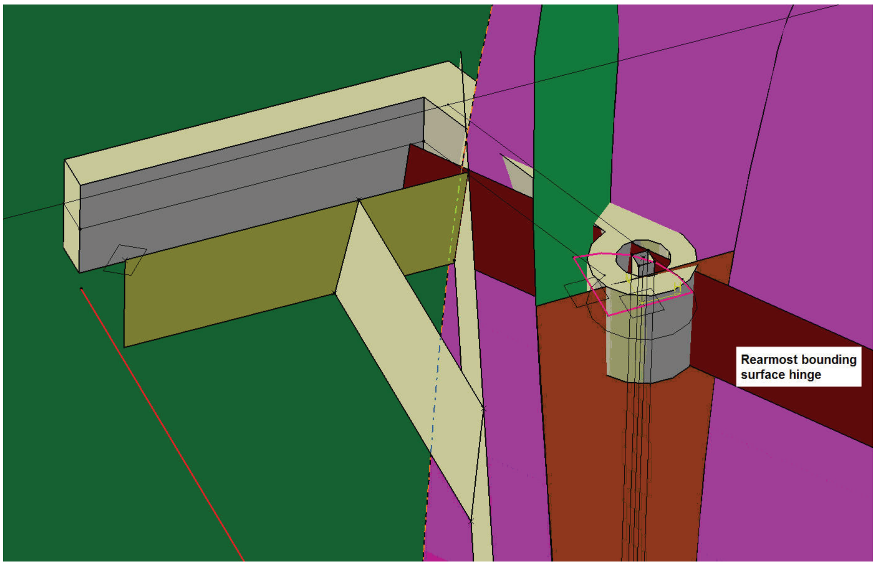


Fig. A.2.10 Design of rearmost bounding surface hinge

The design principle is similar to the design principle explained above. As the theory of the design principle has already been described in detail the description of the design of the two rearmost bounding surfaces for the hinges are made in a shorter form for one hinge (see figure A.2.10). First of all the possible contact surface of the door flange with the hinge is designed. An offset of the contact surface is designed according to the variant of the door flange (with or without crash profile). The theoretical corner of the door is projected in Y-direction onto the offset surface. The length of the theoretical front corner of the door is trimmed according to the height of the offset surface. Trimmed front corner and projection of front corner are connected by a swept surface. From the swept surface another swept surface is designed under an angle of 90° - door opening angle along the trimmed front corner and intersected with the offset of the contact surface hinge. From the XZ-plane another swept surface is designed along the intersection curve under the complementary angle ((180° - door opening angle): 2) as described above. The last swept surface is the rearmost bounding surface collision door corner with hinge.

A.2.3.3 Foremost Bounding Surface Collision Door Outer Panel with Front Wing

With the simple design method mentioned in the first paragraph the early Class-A surface is turned around the hinge axis (see figure A.2.1). The intersection curve of the turned and unturned Class-A surfaces is the rearmost bounding curve for the positioning of the door corner in the area of hinge axis. When a door corner is chosen behind this bounding curve the outer panel of a front door will collide with the rear corner of the front wing (compare with foundation).

In this detailed design method the positioning envelope for the hinge axis must be defined. Therefore two different foremost bounding surfaces must be designed and evaluated:

- Foremost bounding surface collision door outer panel with front wing
- Foremost bounding surface turn by corner door at front wing

As visualised in figure A.2.8 the intersection point between opened and closed door (Class-A surface) lies 0.13 mm in front of the theoretical corner of the door under consideration of 70° door opening angle, minus tolerance of 0.8mm for the door gap and clearance 2mm between outer panel door and corner front wing. When hinge axis points are defined with any distance to this point and the Class-A surface of the closed and opened door kite geometries (deltoids) appear (see figure A.2.8). The points are all positioned on the centre line of the kite while the Class-A surfaces and perpendicular lines from the hinge points onto the Class-A surfaces define the outer lines of the deltoids. Closed and opened door (Class-A surface) and the centre line show the opening angle plus two complementary angles (e.g. 70° door opening angle plus 2 x 55°, 80° door opening angle plus 2 x 50°). As soon as the door opening angle changes the distance between theoretical corner and intersection point changes as well as the complementary angles (see figure A.2.11).

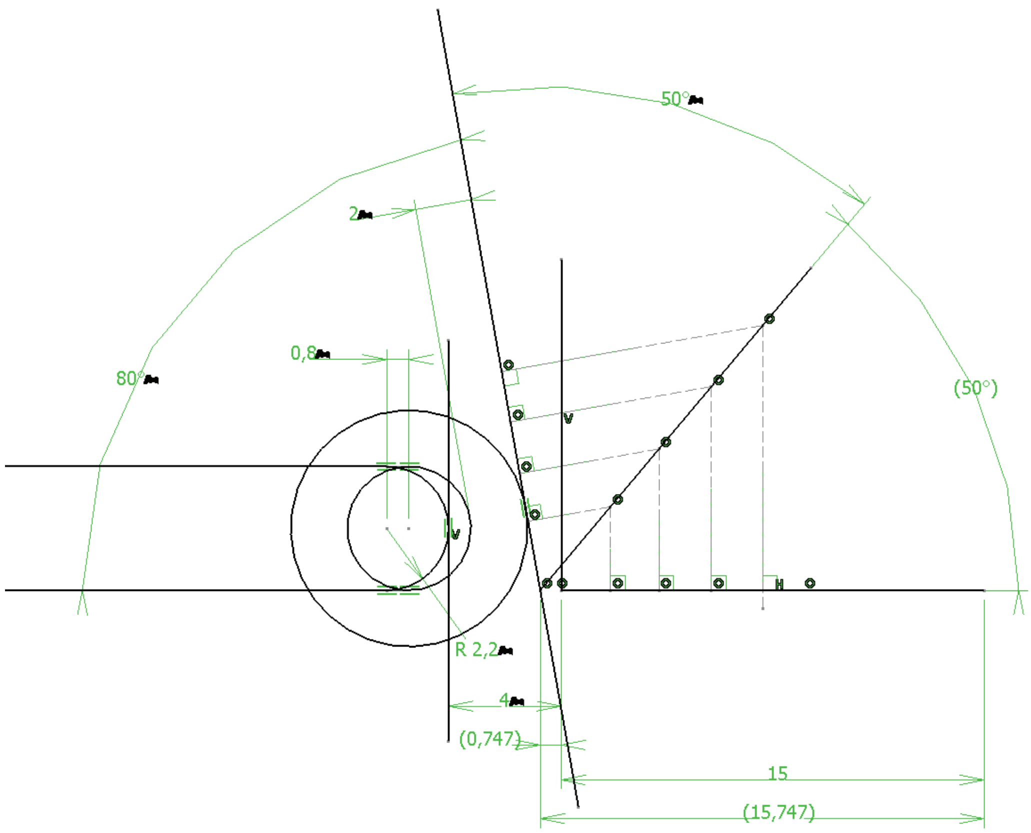


Fig. A.2.11 Reference section for the control of angle relations and distances

In the CAD-model the first step is to develop a curve parallel to the theoretical door corner with the distance taken from the reference section for the control of distances and complementary angles (e.g. for 70° opening angle = 0.13mm (see figure A.2.8), for 80° opening angle = 0.747mm (see figure A.2.11)). Starting from the Class-A surface of the closed door a swept surface is designed along the parallel curve according to the complementary angle of the reference section. The result is the foremost bounding surface which defines one of two different borderline positions of the hinge axis. If a hinge axis lies on this surface the parameters for the collision between door outer panel and rear corner of the front wing are fulfilled. If the hinge axis is located behind the surface no collision problems will occur.

A.2.3.4 Foremost Bounding Surface “turn by” Corner Door at Front Wing

The development for the foremost bounding surface (turn by) is performed on basis of bitangential circles (see figure A.2.12). The design is carried out under support of several horizontal sections. The centre points of the bitangential circles define the foremost bounding curve in every section. The radii of the circles which are tangential to the radius of the door flange as well as tangential to the radius of the front wing define the length of the foremost bounding curve and thus the width of the foremost bounding surface for the critical case of turn by.

To take into account even very complex Class-A surfaces a lot of horizontal sections with short distance have to be developed to represent the outer surface of the door. In the CAD-model the height area is defined and supplied with 31 height planes with a maximum distance of 25mm each. The height area is designed on basis of the highest and the lowest points of the reference surface. Is the reference the complete Class-A surface of the sidewall extreme points for the height area must be chosen on waist line and lower corner of the door to delimit the Class-A surface.

As the design of the bitangential circles and the resulting foremost boundary curve is identical for all horizontal sections the design is carried out only once in a reference section on the XY-plane. First of all in this reference section the details of the door gap are designed (see figure A.2.13). Under consideration of the negative tolerance for the door gap (4mm – 0.8mm) and the “turn by clearance” of the door corner (+1mm) bitangential circles between R6 and R100 are designed and associated with the radius of the door flange and the radius of the front wing under support of the geometrical constraint “tangential”. The centre curve of all radii is the foremost bounding curve for the critical case “turn by”. For a simplified design the centre point of the smallest radius and the centre point of the largest radius could be connected by a straight line. The reference section is completed by reference X- and Y-distances for the start point of the foremost bounding curve (see figure A.2.12).

Firstly each of the 31 height planes is intersected with the theoretical front corner of the door. Relative to this point the start point for the foremost bounding curve is designed. The foremost curve of the reference section is shifted to the particular start point. As the design intend is the same for each of the 31 horizontal sections, the modelling is only conducted for the first height plane and replicated for all other 30 height planes. On basis of the 31 foremost bounding curves the foremost bounding surface for the critical case of „turn by" is designed (see figure A.2.14).

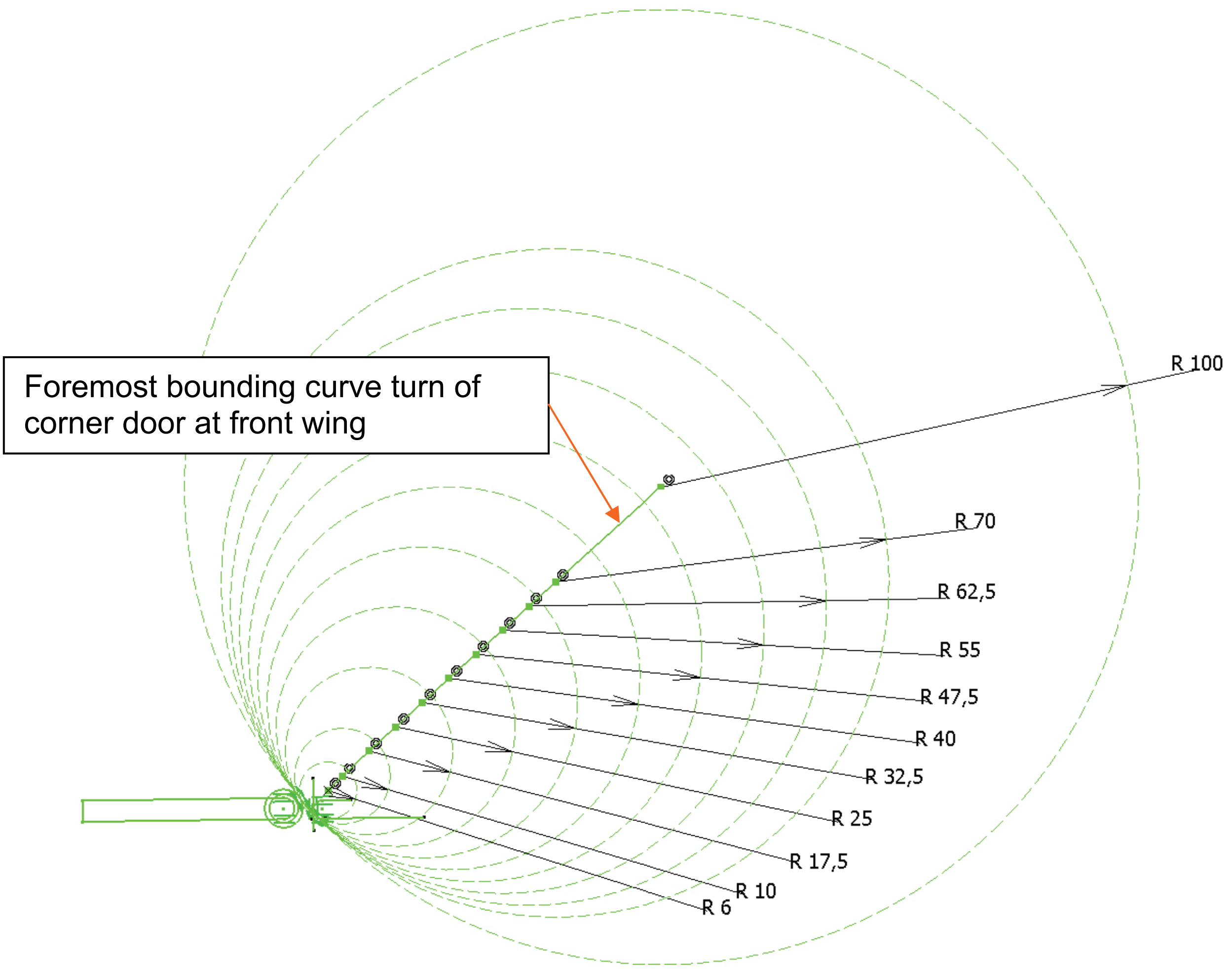


Fig. A.2.12 Reference section: Design of foremost bounding curve by bitangential circles

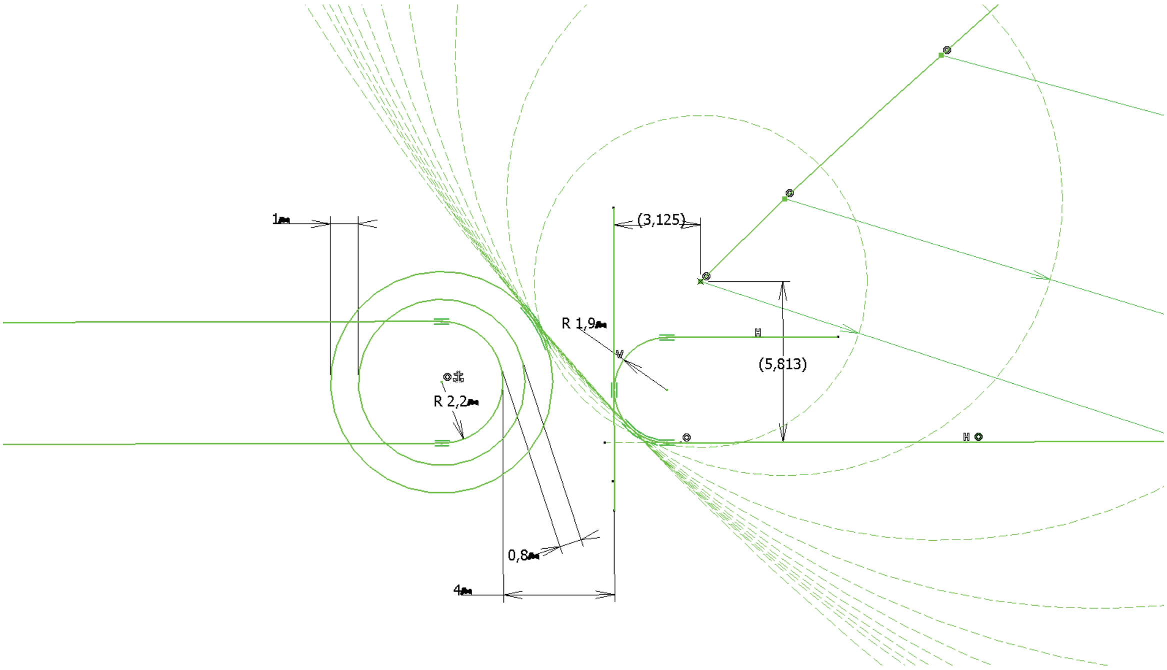


Fig. A.2.13 Reference section: Dimensional detail in door gap area

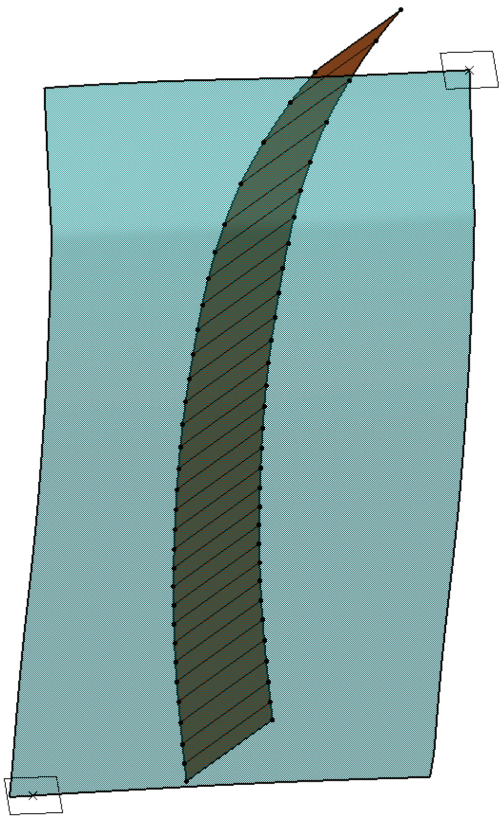


Fig. A.2.14 Several “Foremost bounding curves” define the “Foremost bounding surface „turn by“ and Class-A surface”

A.2.3.5 Selection Foremost Bounding Surface critical Case “turn in” or “turn by”

Depending on the position of the reference door gap defined, the CAD-model must chose between “foremost bounding surface collision door outer panel with front wing” and “foremost bounding surface turn by corner door at front wing”. If the door gap is positioned relatively far back the door outer panel will collide with the front wing.

As both bounding surfaces are designed in the CAD-Model a knowledge ware rule (see chapter A.2.6) chooses that surface which lies in X-direction the furthest back and narrows the positioning envelope for the hinge axis the most.

A.2.3.6 Outmost Bounding Surface Hinge Axis to Class-A Surface

To reduce the redundant forces on door and pillar a maximum moment arm should be provided by the distance between the two hinges of the door (in the CAD-model 320mm). On the other hand will a big distance between the hinges increase the distance between the hinge axis and the curved Class-A surface (see figure A.2.15). This large distance will require additional space for the door flange turning into the sidewall while the door is opened. This arrangement does not allow enough space for the body structure in the finite thickness of the sidewall.

When the hinge axis is inclined around X-axis the distance between Class-A surface and hinge axis could be reduced. To avoid the loss of the ease of use while the door is opened and closed, the angle of inclination should not be increased more than 1.5°.

The outmost bounding surface defines the borderline position between the straight hinge axis and the curved Class-A surface (see figure A.2.15). This distance is influenced by the distance between the hinges and the height of the hinge joints. The outer radius of the hinge joint plus the adjustment travel between hinge and door define an offset needed relative to the hinge axis. Along the door flange the sheet thicknesses of the inner and outer door panel plus a minimum distance for the cataphoretic painting (min. 3mm) plus the minimum distance between the door inner panel and the hinge joint define an offset needed relative to the Class-A surface.

Fig. A.2.15 Turned in door with crash profile, hinge and pillar (Fischer, 2006)

For the design of the outmost bounding surface first of all an offset of the Class-A surface is designed according to the minimum distance required between Class-A surface and hinge joint, as explained above. This offset is moved horizontally to the inside of the car by the outer radius of the hinge joint plus the travel adjustment. The reference surface of Class-A is bounded by the theoretical door corner. Relative to the XZ-plane a new plane is designed at an angle about the X-axis according to the angle of inclination expected for the hinge axis.

On the theoretical door corner the outmost and foremost point is modelled. In this point a normal plane to the XY-plane is defined. This foremost plane on the door corner borders the outmost bounding surface to the front. Parallel to the foremost plane two additional planes are defined with distances of 40mm. These planes define the centre and the rearmost border of the outmost bounding surface. The three planes are intersected with the inclined plane and the offset of the Class-A surface, which has been moved. On each of the three normal planes a *positioned Sketch* with a straight line is defined. This straight line has the distance of the length between hinges plus the height of both hinge joints. It is parallel to the intersection line of the normal plane and the inclined plane and its end points are coincident with the intersection curve of the moved offset of the Class-A surface. The three modelled and positioned straight lines define the outmost bounding surface (see figure A.2.16).

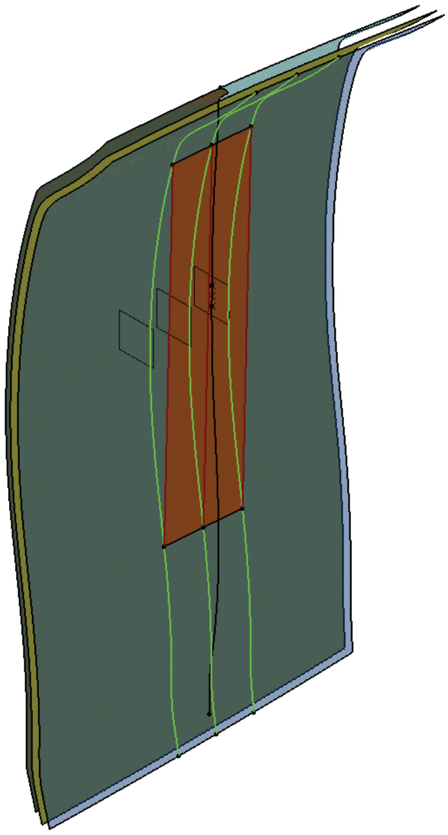


Fig. A.2.16 Design of outmost bounding surface by three secants on the Class-A surface, that has been moved insight

A.2.4 Tolerance Envelope for the chosen Hinge Axis

On the endpoints of the centre line of the outmost bounding surface two planes are defined parallel to the XY-plane which are the bases for the hinge positioning. On the lower plane the start point of the chosen hinge axis is defined which must be adjusted manually in X- (H-) and Y- (V-) directions. On the upper plane the end point of the chosen hinge axis is defined which must be adjusted manually in X- (H-) and Y- (V-) directions. By (Knowledge Based Engineering) KBE the endpoint can be adjusted by an optimising algorithm to calculate target values for the angles of inclination around the X- and Y-axes. Start and end point of the hinge axis are connected by the hinge axis.

Based on the fact that the maximum angles of inclination are 1.5° around X-axis at the top to the inside and 1.5° around Y-axis at the top to the back a circular sector is designed on the upper plane to support the manual adjustment of the chosen hinge axis relative to the positioning envelope. The centre point of the circular sector is the projection of the start point onto the upper plane. The circular sector is bounded by two lines in +X- and +Y-directions and a circle. The radius of the circle is calculated in the CAD-model on basis of a trigonometric function by the angle of inclination and the distance of lower and upper plane (see figure A.2.10). If the angles of inclination around X- and Y-axes are both 1.5° the position of the end point is the intersection point between circle and bisecting line of X- and Y- direction in a projection perpendicular to the lower plane.

Lower and upper plane, hinge axis, outer pillar surface and explicit parameters (see figure A.2.17) define the available space for the hinges. In the CAD-model hinges are defined because the contact surface between door flange and hinge is needed for the design of the positioning envelope. The designed hinges can at any time be replaced by Carry-Over-Parts (COP). Therefore the new hinges must be placed according to the available space and the hinge axis. The contact surfaces of the designed hinges must be replaced by the contact surfaces of the new hinges. The explicit parameters of the old hinges must be changed according to the parameters of the new hinges as the explicit parameters are also used for the design of the positioning envelope. If the explicit parameters for an angle of inclination or the tolerance area of the hinge axis in X- or Y-direction are changed, this angle or this tolerance offset must also be changed during optimisation.

In the middle of the hinge axis a normal plane to the hinge axis and four different lines perpendicular to the hinge axis are modelled. This plane is then used for a control section. The four lines are used to check whether the chosen hinge axis lies in or outside the positioning envelope.

- Parameter_Scharniere (Parameters_hinges)
  - `Zielwinkel_Neigung_SA_um_Y (Target_angle_inclination_hinge_axis_around_Y)` =1,5deg
  - `Zielwinkel_Neigung_SA_um_X (Target_angle_inclination_hinge_axis_around_X)` =1,5deg
  - `Mindestabstand_Strak_Scharniergewerbe (Minimum_distance_Class-A_hinge_bearing)` =9,5mm
  - `Verstellweg+Radius_Scharniergewerbe (Adjustment_travel+radius_hinge_bearing)` =8mm
  - `Hoehe_Scharniergewerbe_unteres_Scharnier (Hight_hinge_bearing_lower_hinge)` =17,5mm
  - `Hoehe_Scharniergewerbe_oberes_Scharnier (Hight_hinge_bearing_upper_hinge)` =17,5mm
  - `Mindestabstand_Scharniergewerbe (Minimum_distance_hinge_bearings)` =320mm
  - `Laenge_Anlageflaeche_Scharnier_Saeule (Lengths_contact_surface_hinge_pillar)` =60mm
  - `Dicke_Scharnier_saeulenseitig (Thickness_hinge_pillar_side)` =6mm
  - `Anstellwinkel_Kroepfung (Angle_of_inclination_bent_portion)` =70deg
  - `Dicke_Scharnier_Abstellung (Thickness_hinge_bent_portion)` =8mm
  - `Radius_Scharnierbolzen (Radius_door_nail)` =4mm

Fig. A.2.17 Explicit parameters for the design of hinge axis and hinges

For the design of the tolerance envelope explicit parameters are defined for the tolerance areas in +/- X and Y-directions. Under support of this parameters four borderline positions of the hinge axis are defined which describe a rectangular envelope.

A.2.5 Inspection of Door hang-out

During manufacturing normally the closures are fixed to the body in white for the adjustment and painting processes. For the final assembly the closures are removed and then assembled on separate assembly lines. Often the bifid joints of the hinges are disconnected to enable to reproduce the door position after final assembly. In most cases the hinge bolts are dismounted and the door is removed in a perpendicular direction to the hinge axis. In some cases the doors are removed for the hang out in the direction of the hinge axis and as soon as the upper parts of the hinge joints are clear from the hinge bolts the door is removed perpendicular to the hinge axis. In the CAD-model it is checked whether the door hang out for this case is possible without interference between the partially opened door and the front wing.

In the CAD-model the trimmed Class-A surface of the door is completed by the door flange. Therefore one of two true reference sections (crash or hemming profile) must be selected by KBE and moved to the theoretical door corner under support of local coordinate systems to be designed in the reference sections as well as on the theoretical door corner.

After the selection of the profile variant under support of a rule (knowledge ware) the prismatic door flange is designed. The designed door is opened partially and lifted up into the hang out position (see figure A.2.18). A check (knowledge ware) measures the distance between opened and lifted door and front wing and compares this distance with the explicit parameter for the minimum distance door hang out.

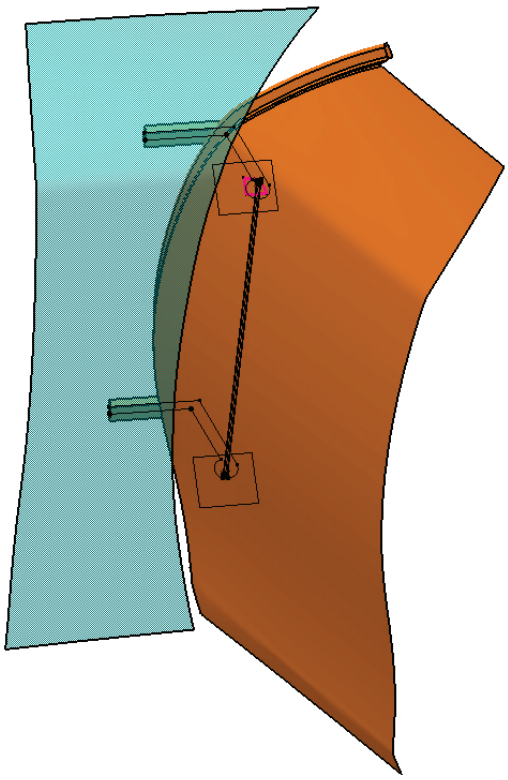


Fig. A.2.18 Partially opened and lifted up door in hang out position

A.2.6 Application of Knowledge Based Engineering (KBE)

For the layout of the positioning envelope and the examination and adjustment of the chosen hinge axis rules, checks and optimisations are defined.

A.2.6.1 Rules

For the control of the CAD-model three rules are defined:

- Selection rule of profile variant of door flange (hemming or crash profile) to control an explicit parameter for a distance
- Selection rule between foremost bounding surface turn by corner door at front wing and foremost bounding surface collision door outer panel ant front wing
- Selection rule of profile variant of door flange (hemming or crash profile) for the design of the prismatic door flange

Selection rule of profile variant of door flange (hemming or crash profile) to control an explicit parameter for a distance:

Associated rule and parameters are placed in the main relations and parameter set of the structural tree.

An explicit parameter with switch-function *(parameter type "String")* allows choosing between two variants. In this case the parameter is called "crash profile" and with "Yes" and "No" the user can choose between crash and standard hemming profile (see figure A.2.19). According to the switch a second parameter (parameter type "Length") gets offset information from one of the two reference sections. On the XY-plane the two reference sections of the door flange with and without crash profile are designed. In both sections the distances between theoretical corner door and outer pillar surface including tolerance for the gap and clearance are dimensioned. Under support of the rule the parameter *"Distance_theoretical_corner_door_vs_ pillar_geometry"* is applied with the correct offset. The offset is measured by the rule with *Measure Distance (Body\Body)* between two output elements (3D geometry) of the chosen reference section. In this way instead of the complete design of the door flange the distance between theoretical corner door and pillar geometry are taken into account which allows a simpler and update safe approach.

Selection rule between “foremost bounding surface turn by corner door at front wing” and “foremost bounding surface collision door outer panel ant front wing”:

The rule *Selection_foremost_bounding_surface* is placed in the structural tree in the set of the positioning envelope directly behind the design of the two different foremost bounding surfaces in a set called *Selection_foremost_bounding_surface_ critical _case*. This placement was chosen to allow the user a better traceability.

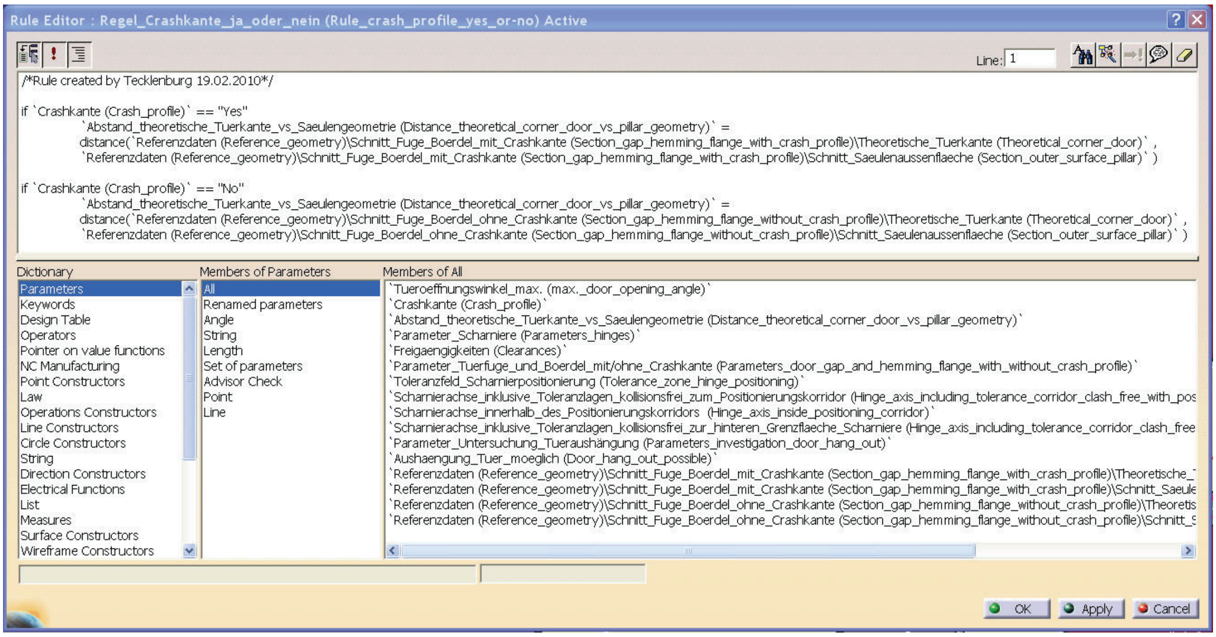


Fig. A.2.19 Control of a distance parameter according to the profile variant

Depending on the position of the foremost bounding surfaces the rule selects between foremost bounding surface collision door outer panel with front wing and foremost bounding surface turn by corner door at front wing. The rule selects the foremost bounding surface which is the furthest to the back and narrows the positioning envelope the most.

Therefore it is necessary to design some auxiliary geometry: Both foremost bounding surfaces are intersected with the rearmost bounding surface. The two intersection curves are projected onto the XZ-plane. The rule selects the foremost bounding surface whose projected intersection curve has in X-direction the bigger offset distance to the YZ-plane.

For the development of the rule first of all an explicit parameter *(parameter type surface)* is defined. This parameter (this surface) is used for the following design of the positioning envelope. The rule measures the distance between the projected curves and the YZ-plane and allocates that foremost bounding surface to the parameter of type surface whose intersection curve lies the furthest to the back (see figure A.2.20).

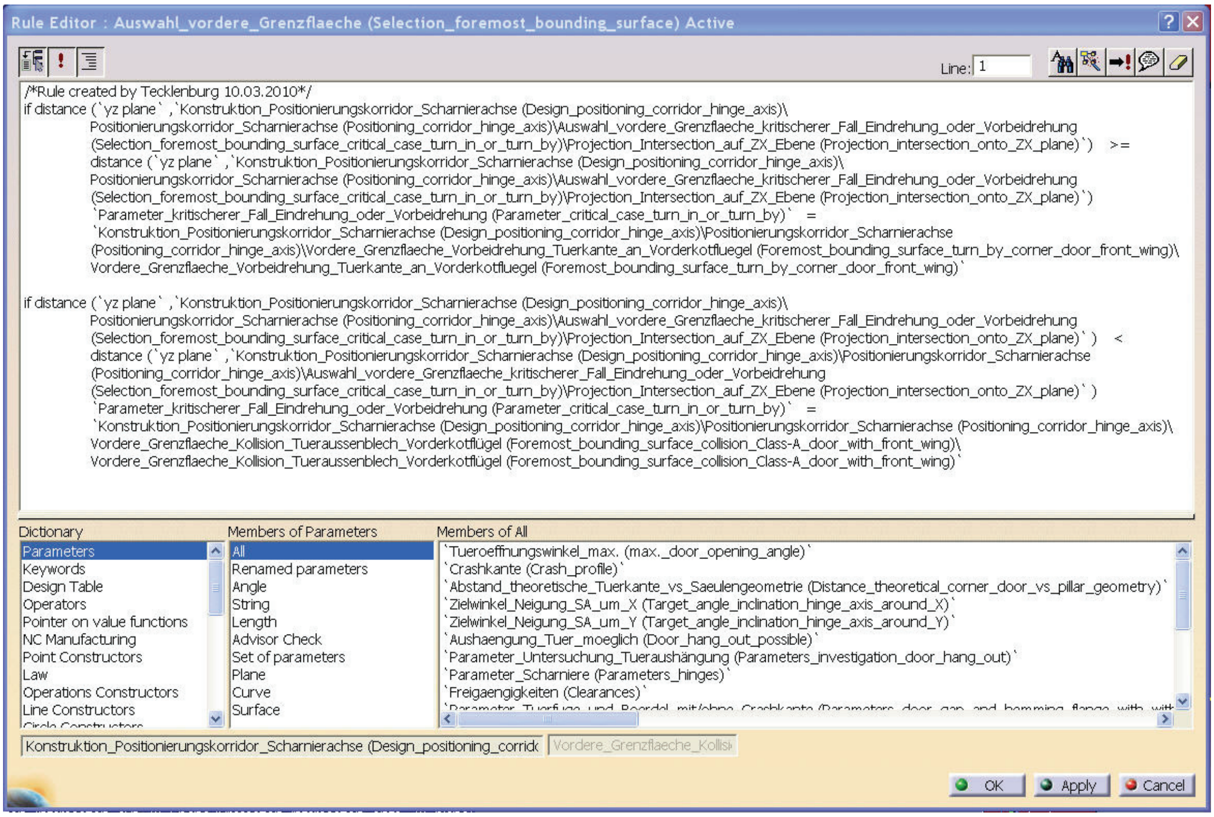


Fig. A.2.20 Selection between two different foremost bounding surfaces

Selection rule of profile variant of door flange (hemming or crash profile) for the design of the prismatic door flange:

The rule *Selection_profile* is placed in the structural tree in the set of the "investigation door hang out" in a set called *Selection_profile.* This placement was chosen to allow the user a better traceability.

For the investigation whether the defined door gap allows a door hang out the Class-A surface must be completed with the prismatic door flange. One section must be selected for the surface design from two reference sections defined in the CAD-model.

An explicit parameter (parameter type "String") allows choosing between two versions. The parameter "Crash profile" which allows choosing with "Yes" or "No" between a crash profile and a standard hemming profile is already known from the first rule.

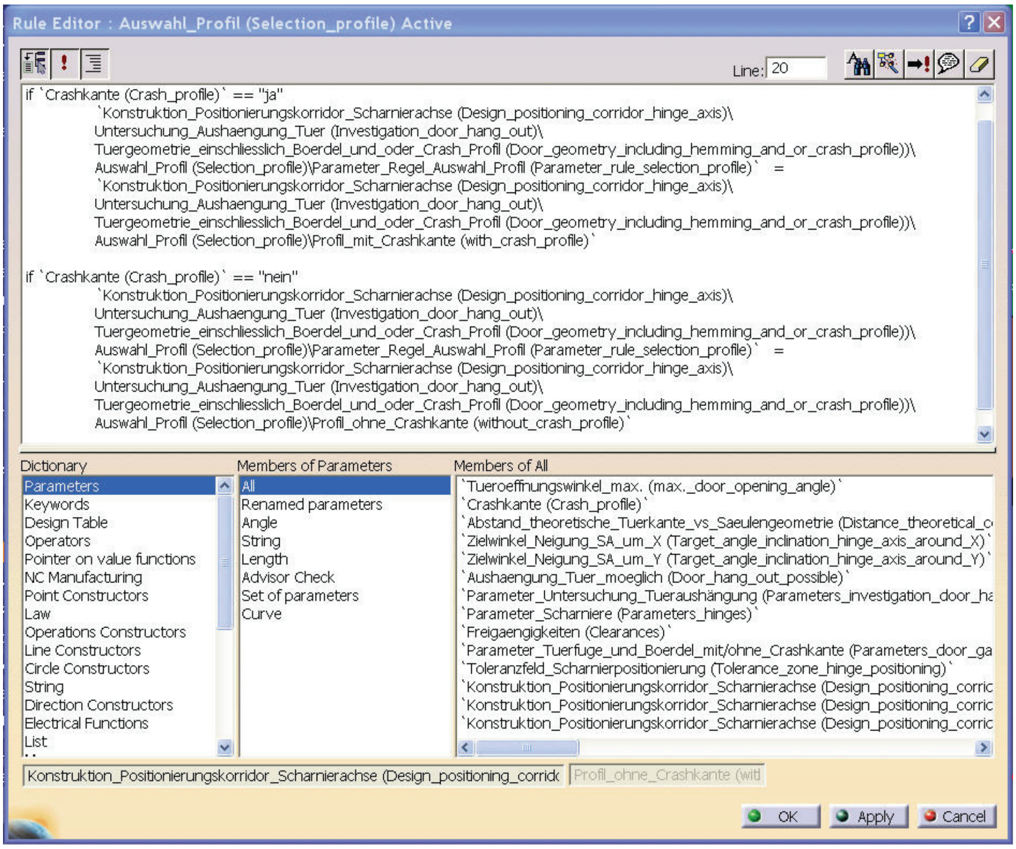


Fig. A.2.21 Selection of the true profile for the design of the door flange

For the development of this rule first of all an explicit parameter of type „curve" is defined. This parameter (this curve) is the profile which is used for the following design of the prismatic door flange. For the design the sketch outputs of both reference profiles are shifted to a plane perpendicular to the theoretical door corner. Is the setting of the switch “Yes” the rule allocates the shifted output “crash profile” to design the flange surface, is the setting “No” the shifted output of standard hemming flange is chosen (see figure A.2.21).

From the positioned true profile the flange surface is designed. The completed door is partially opened by the parameter *Opening_angle_door_at_hang_out.* Under support of a KBE check the distance between door and front wing is inspected.

A.2.6.2 Checks

Fig. A.2.22 Check door hang out: LH: door hang out possible, RH: door hang out not possible

In the CAD-model four checks are integrated. The checks help the designer to find a reliable hinge axis position and answer the question whether the reference door gap allows a door hang out:

- Check whether door hang out for dismounting is possible.
- Check whether the hinge axis with tolerance envelope intersects with the positioning envelope.
- Check whether the hinge axis is positioned inside the positioning envelope.
- Check whether the hinge axis with tolerance envelope intersects with the rearmost bounding surfaces of the hinges.

Check whether door hang out for dismounting is possible:

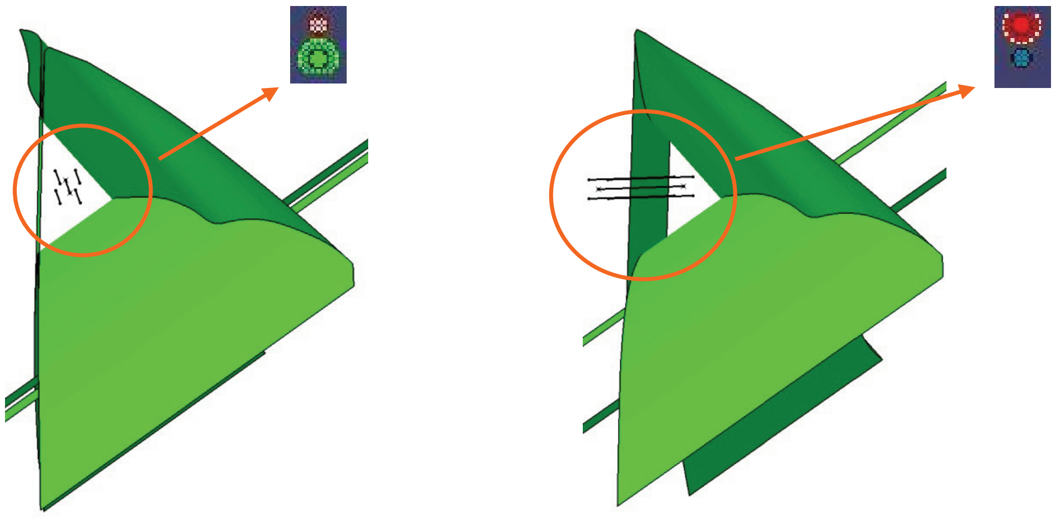


Fig. A.2.23 Check whether hinge axis including tolerance envelope intersects with positioning envelope

The Check *Door_hang_out_possible* controls whether it is possible to dismount the door under a defined door opening angle and whether the distance between opened and lifted up door and front wing does not fall below the defined clearance. The hang out travel in the CAD-model is half the height of the hinge joint plus 2mm clearance (see figure A.2.22).

Check whether hinge axis with tolerance envelope intersects with positioning envelope:

The check *Hinge_axis_including_tolerance_envelope_clash_free_with_positioning_ envelope* controls whether the hinge axis including its tolerance positions (designed by lateral displacements of the hinge axis in X- and Y-directions) do not clash into the positioning envelope (see figure A.2.23). If the check shows a clash the chosen hinge axis must be moved manually. As it is possible that the check does not show a clash because the hinge axis lies completely outside the positioning envelope another check is necessary to ensure whether the hinge axis is positioned inside or outside the positioning envelope.

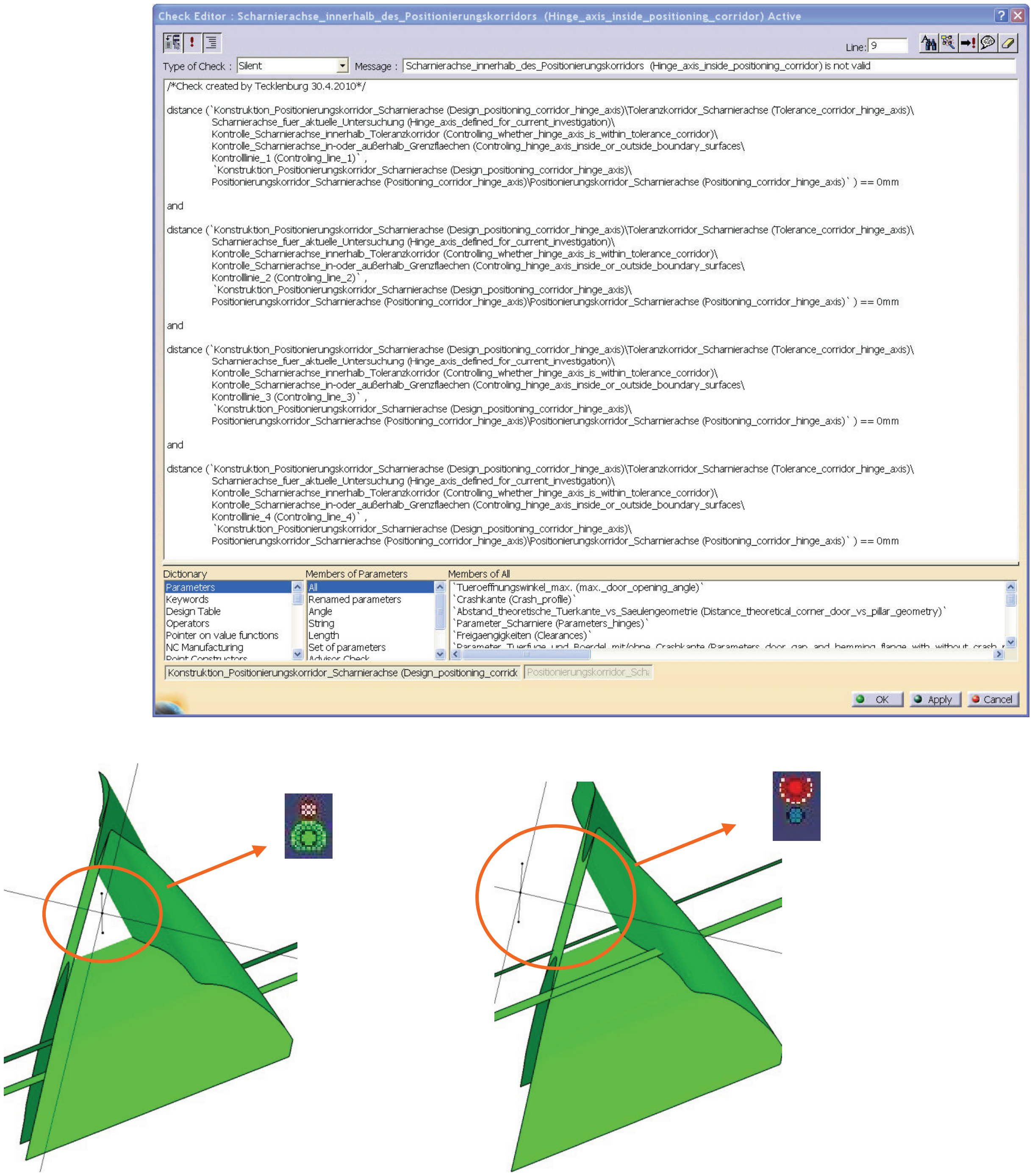


Fig. A.2.24 Check whether the hinge axis is positioned inside the positioning envelope

Check whether the hinge axle is positioned inside the positioning envelope:

The check *Hinge_axis_inside_positioning_envelope* controls whether the centre of the hinge axis is completely positioned inside the positioning envelope. Four auxiliary lines are defined in the centre point perpendicular to the hinge axis. Only if all four auxiliary lines intersect the positioning envelope the centre of the hinge axis lies inside the positioning envelope. Only if both, this check and the check *Hinge_axis_including_tolerance_envelope_clash_free_with_positioning_envelope* show a green traffic light it is guaranteed that the complete length of the hinge axis and its tolerance positions are positioned clash free inside the positioning envelope (see figure A.2.24).

Check whether the hinge axis including tolerance envelope intersects with the rear-most bounding surfaces of the hinges:

Similar to the check which controls the door hang out this check measures the minimum distance between the tolerance envelope of the hinge axis and the rearmost bounding surfaces of the hinges. These bounding surfaces additionally narrow the positioning envelope for the hinge axis.

A.2.6.3 Optimisations

Optimisation angles of inclination of hinge axis within the positioning envelope:

The *Optimizer* is part of the CATIA-workbench *"Product Engineering Optimizer"*. The *Optimizer* is working on base of different mathematical algorithms such as the gradient algorithm or the stochastic simulated annealing algorithm. Under support of the *Optimizer* the following problems can be solved: A minimum or maximum of a target value can be calculated under consideration of several conditions and constraints.

For the definition of an optimisation the feature "*Optimize*" must be chosen. The pull down menu of the feature shows three tabs: *"Problem"*, *"Constraints"* and *"Computations results"*.

In the *Problem*-Tab (see figure A.2.25) the user decides which kind of optimisation should be conducted (identification of a minimum or maximum target, alteration of target value or explicit parameter by constraints). In the CAD-model the target values for the angles of inclination around X- and Y-axes are defined. The *Optimizer* offers five different calculation algorithms. The two most important are the *"Simulated Annealing Algorithm"* and the *"Gradient Algorithm"*.

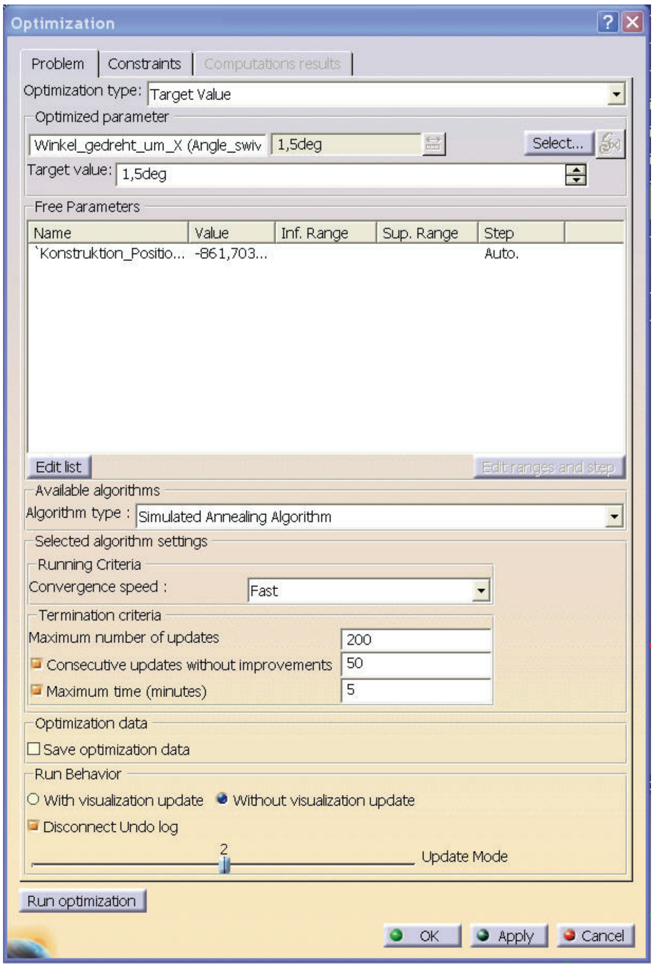


Fig. A.2.25 Problem-Tab

The *"Simulated Annealing Algorithm"* is a global stochastic search algorithm. Two executions of this algorithm one after another probably do not show the same result (Dassault Systémes, 2009). In the CAD-model this algorithm is chosen because of the complex shape of the positioning envelope. The algorithm is used several times shifting between optimisation around X- and Y-axis.

The *“Gradient Algorithm”* searches for a local minimum or maximum. Based on the calculation of the local gradient of the target function the algorithm uses a parabolic approximation and jumps step by step to its minimum or uses one iterative exponential step downward in the direction towards its minimum (Dassault Systémes, 2009). The minimum found by this algorithm depends on the initial value. This means that the result is a local minimum. The local minimum is not mandatory the global minimum of the target function. This is the reason why this algorithm was not chosen. The calculation settings can be specified in the tab precisely. The convergence speed, the number of updates, the maximum number of updates without improvement and the calculation time can be defined.

When the option *“Save optimization data”* is chosen the third tab *“Computations results”* is activated. An Excel-file will be produced and saved at a defined path. The tab communicates with this file. In case of a quick calculation optimisation data are not saved in the CAD-model.

The bottom *“Run Optimization”* starts the calculations.

The feature *“Without Update Visualization”* improves the calculation speed as none of the parameter optimisations will be visualised in the design model. When the optimisation is finished the update has to been done manually.

In the *Constraints*-Tab (see figure A.2.26) the controlling conditions for the calculation are defined. In the CAD-model the minimum distance between hinge axis and positioning envelope / sqrt (0.5) > 2.5 mm is defined as *“Constraint”* according to the explicit parameter for the tolerance range for the hinge axis. The tolerance range of the hinge axis in X- and Y-direction is 2.5 mm. Multiplied with sqrt (0.5) result the radius for the tube of minimum distance. The feature *“New”* allows to define the constraint. In the *Constraints*-tab a value is shown which describes the difference between the constraint and the measured distance in the CAD-*model (Distance to* *satisfaction)*. The user can define the weighting for the value. Can the calculation not satisfy the weighted value a red traffic light will be shown.

When the option *"Save optimization data"* is chosen the third tab *"Computations results"* is activated. After the calculation is finished the user has the opportunity to visualise and sort the calculation results either chronological or according to a rated value. Every single intermediate result can be chosen and used for a recalculation. In the CAD-model optimisation results are not saved on behalf of an expected quicker calculation.

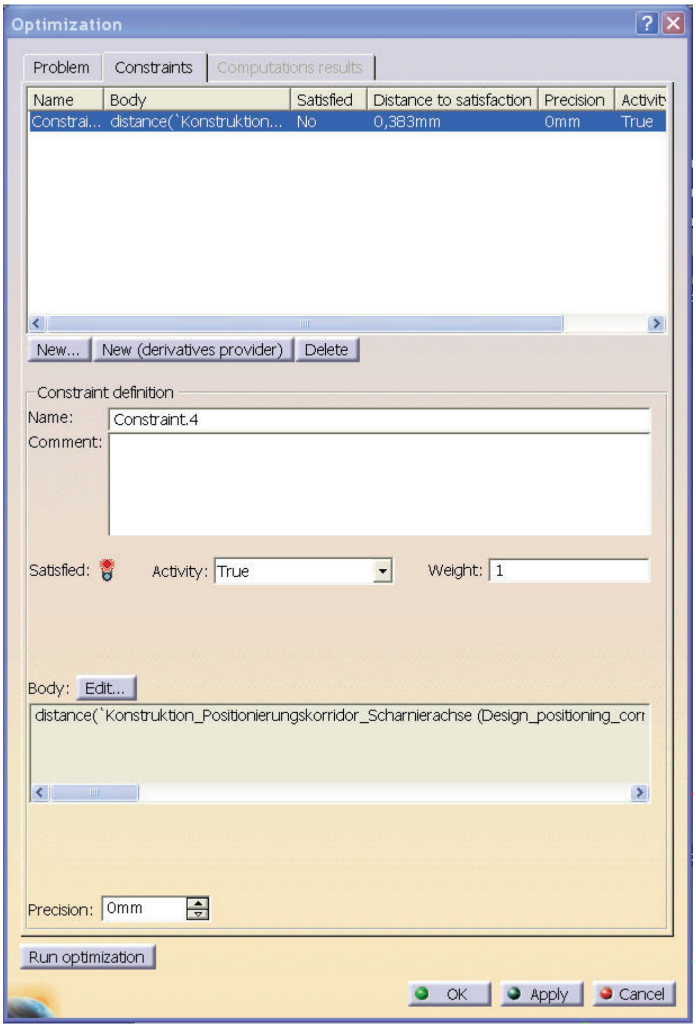


Fig. A.2.26 Constraints-Tab

In the CAD-model the angles of inclination around X- and Y-axis are optimised iterative changing between optimisation around X- and Y-axis. Alternatively both parameter could be optimised together using the function $U = \sqrt{} X^2 + Y^2$.

A.2.7 Summary

Under support of the CAD-model a positioning envelope for a new optimised hinge axis position of a front door with turn-in arrangement can be developed under consideration of the reference geometries Class-A surface with door gap and outer surface of the A-pillar. The CAD-model provides proposals for the hinge axis and the hinges. Explicit parameters such as door opening angle, target values for the angles of inclination of the hinge axis around X- and Y-axis, dimensions for different door flanges and hinges as well as clearances and tolerance ranges can be adjusted individually. The provided hinges can be exchanged by Carry-Over-Parts (COP).

After the complete exchange of the reference geometries and individual adjustment of the explicit parameters the provided hinge axis must be adapted in the positioning envelope manually in iterative steps. The adaptation is carried out in a view in the direction of the hinge axis. Firstly all the coordinates of the lower start point and the upper end point of the hinge axis are adjusted under support of auxiliary geometry. Checks provide information whether the expected position of the hinge axis inside the positioning envelope under consideration of clearances and tolerances is achieved. The exact positioning of the hinge axis is calculated in the CAD-model under support of an iterative optimisation algorithm.

Under support of this reusable layout the quick definition of the optimum hinge axis position of a front door can be conducted. With small additional effort the CAD-model can also be used for the development of the hinge axis of the rear door.

Appendix 3 Template "From Concept-Section to true Section and Surface Bands" [5,6]

[5] Student Simon Maurer, head of group project team Porsche had the idea for this *Powercopy* in winter semester 2009

[6] Please see the following CATIA models:
2010-01-02_C-Pillar_Test_bench_Prismatic_areas
2010-01-06_POWERCOPY_Development_True_section_and_Surface_bands _Principle_Maurer

A.3.1 Introduction

This *Power copy* **(Figure 8.25)** automatically develops a true (compound) section and surface bands of prismatic part areas on basis of a true most concept section. The true section of the segment is the foundation of every surface band.

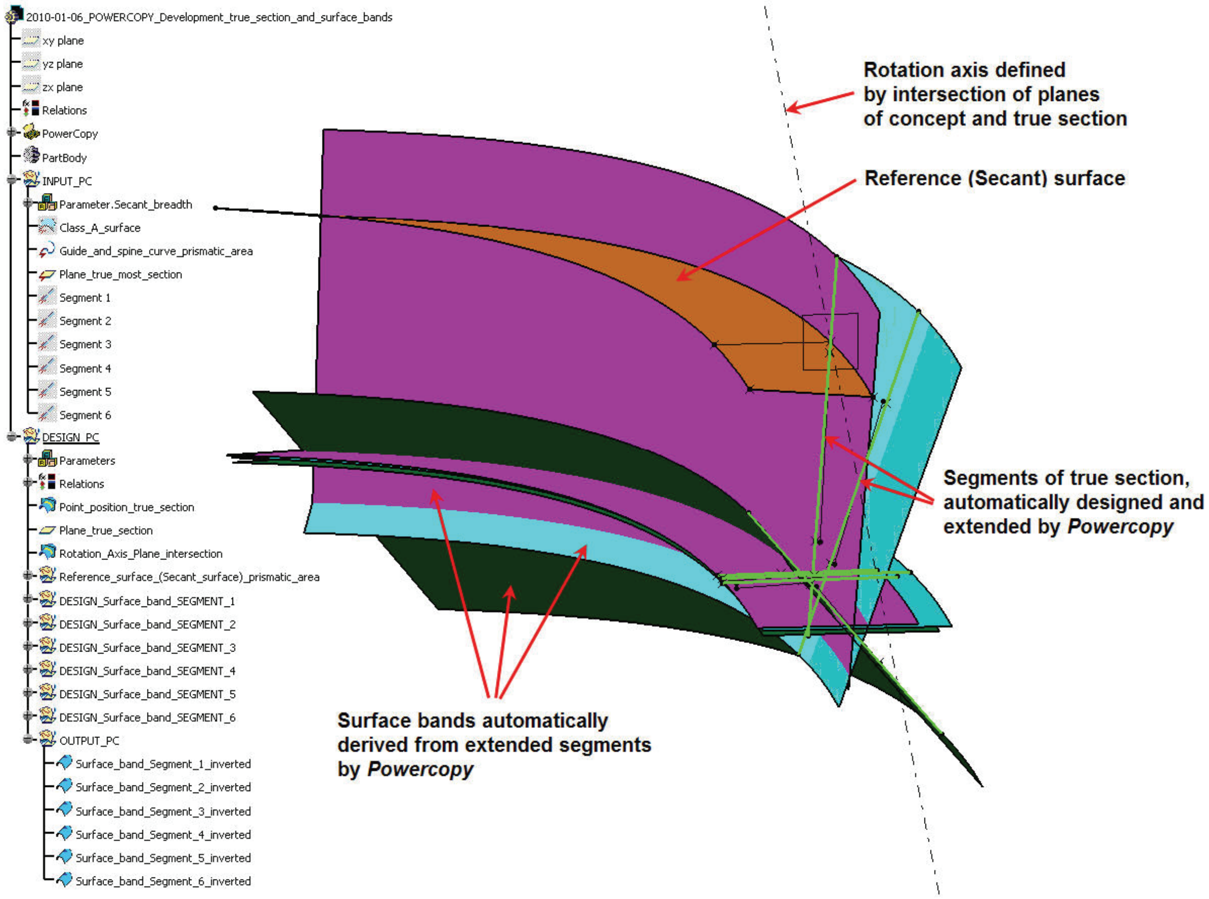


Fig. A.3.1 *Power copy* for the automatically development of a true (compound) section and surface bands from a true most concept section.

Various tests with the *Power copy* have shown that angle variations of ≤1° between reference surface (secant surface) and flanges are possible. As the iterative separate design of the true section and all surface bands derived from this section is very time consuming small dimensional variations are accepted.

A.3.2 Prerequisites for the Application of the *Power Copy*

The basis for the design of assembly areas of automotive bodies are true most concept sections cut perpendicular to the theoretical intersection curve between bounding primary surfaces (e.g. C-pillar: theoretical intersection curve between sidewall and rear wall, upper – see figure A.3.2).

Concept sections deliver the construction method (e.g. shell construction with outer, reinforcement and inner panels) of a body assembly and the surfaces of corresponding parts including all dimensions. For every assembly area (e.g. C-pillar upper) only one concept section is designed. Prismatic areas (areas with constant cross section) are identified in the concept sections. Every prismatic area has its own guide and spine curve.

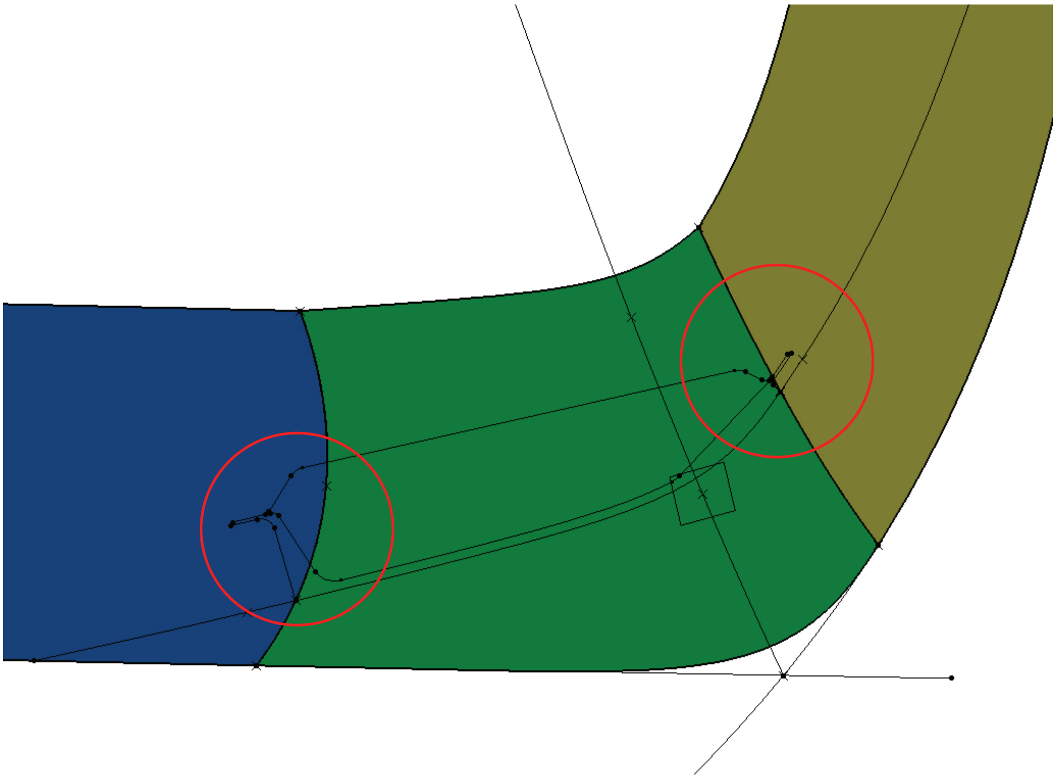


Fig. A.3.2 True most concept section C-pillar with two prismatic areas

Concept sections normally have a common position; the theoretical intersection curve and the guide and spine curve of the respective prismatic area are not parallel. Exceptions are concept sections positioned in grid position (e.g. 0Y) or concept sections cut in an area where both the intersection curve of primary surfaces and the guide and spine curve of the respective prismatic area are parallel. In both cases the concept sections are true sections which can directly be used to derive surface bands. The guide and spine curve of the respective prismatic area and the intersection curve can be in parts or completely parallel for the whole design area.

For these exceptional cases the *Power copy* is not to be used. In the present *Power copy* the angle between the planes for true most and true section is checked while Power copy is inserted (instantiated). Is the measured angle 0° or 180° a warning signal and a comment are generated.

In practice for every prismatic area of a concept section a true (compound) section is designed. The geometrical and numerical parameters of the compound section are linked with the parameters defined in the concept section.

As a general rule joints (transitions) are designed between corresponding assembly areas. Therefore surface bands and in single cases profile bands are derived from corresponding compound sections which are connected by transition surfaces or intersected blunt and filleted in the junction area.

A.3.3 Application of the *Power Copy* to the Concept Design of a Pillar

For every prismatic area the template is used separately (see figure A.3.3). The template allows developing of six surface bands. When it is necessary to develop more than six surface bands the *Power copy* must be used again for the same prismatic area. If the prismatic area has less than six segments, single segments must be used twice and one of the double results must be deleted afterwards.

The template first designs the secant surface (reference surface) on the basis of the Class-A surface and the guide and spine curve. In this case the secant surface is necessary to check the dimensional accuracy inside the template and soon after its application. It is recommended to compare important dimensions defined in concept section with the designs of the template. In the present *Power copy* an intersection line between plane true section and the secant surface is generated to control important dimensions.

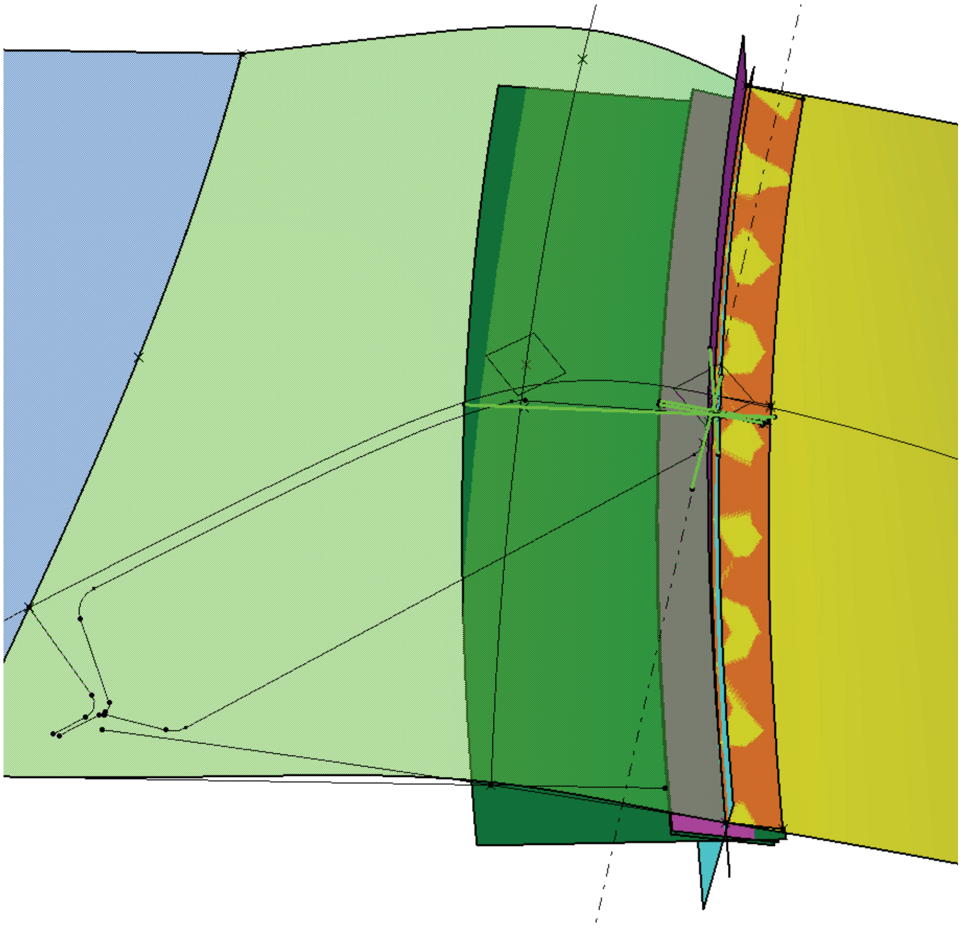


Fig. A.3.3 Application of *Power copy* in the area of window bedding rear window

Every straight as well as curved segment of the concept section is rotated from plane true most section onto plane true section. Therefore the plane of the true most section is intersected with the guide and spine curve of the prismatic area. In the intersection point the normal plane for the true (compound) section is defined.

The intersection line between plane concept section and plane compound section is the rotation axis for the segments of the concept section. Every single segment is rotated onto the true plane of the compound section and is extended curvature continuous at its start and end point. A Boolean parameter asks the user of the present *Power copy* whether the segment is a flange. If the operator indicates the input “true” the segment will not be extended at the end of the flange. If the operator indicates the input “false” the segment is extended on both sides.

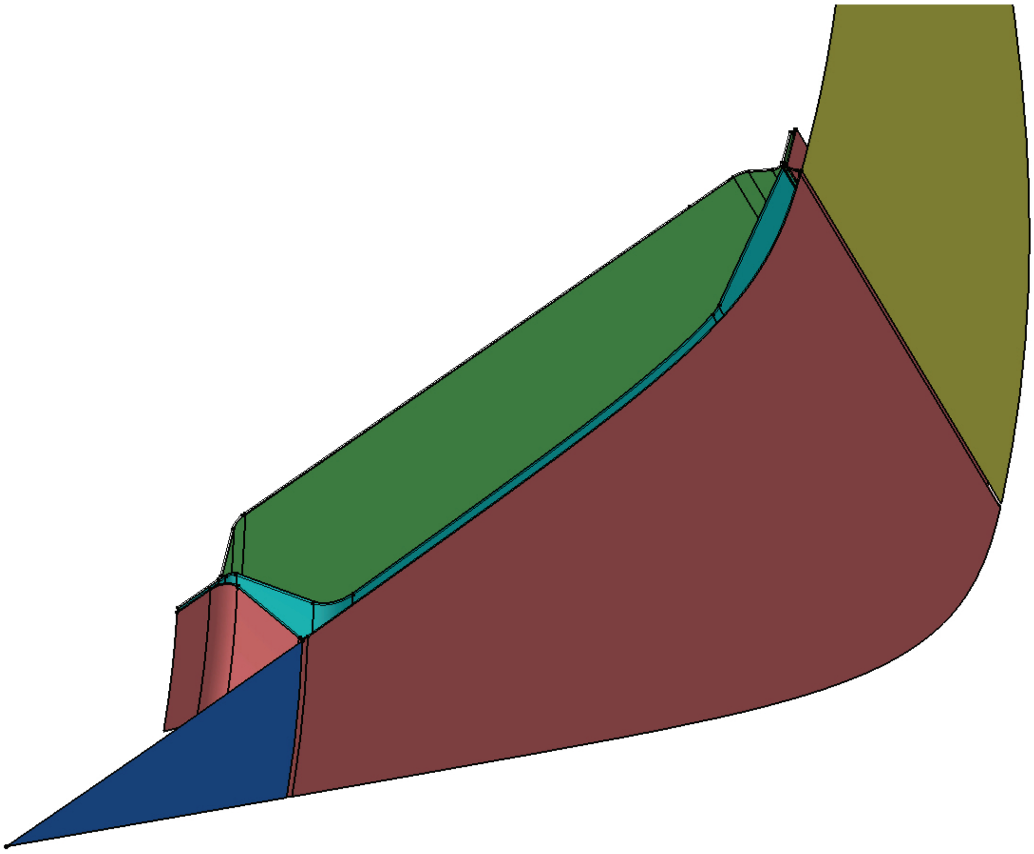


Fig. A.3.4 Control section with view cut through the completed C-pillar design, cut by the plane of the concept section

From every rotated and extended segment a surface band is developed along the guide and spine curve of the prismatic profile area.

The application of the *Power copy* necessitates the check of the directions of curves and planes used. Otherwise it is possible that a true section 180° away from the expected section (mirror image) is generated and surface bands are not in the expected quadrant.

A.3.4 Tips for the progressive Design of Parts

In a real development environment the surface bands are published in the development phase for assembly areas and copied into the adapter of the development phase for single parts. In the new phase joints (transitions) between corresponding assembly areas are designed and the single body parts are derived. In this simplified example only the parts of the upper C-pillar are developed without any junction designs or further detailing (see figure A.3.4).

Appendix 4 Design Guide Lines for collaborative Design Work

The design guide lines carefully developed and approved in several projects are collected in this appendix. The appendix 4 shows design guide lines for team projects and a fully described start model:

A.4.1. Guide Lines

A.4.1.1 Handling of Start Model

In the project structure in file Project-Data / 02Templates / 20 Current the common start model is stored (see chapter 6). The start model has to be opened with "New From". After renaming the model can be saved in the individual file of the designer.

A.4.1.2 Denomination of CAD Files

The following denomination for CATProducts and CAT Parts for Part number, Instance name and file name is mandatory:

KOB_AUDI_ADA___DENOMINATION_OF_PART________________ABC

The denomination of parts and assemblies exactly has 50 characters. The denomination keeps always the same and is never changed.

- KOB Abbreviation of lecture (Konstruktion von Baugruppen = Design of assemblies), 3 characters
- AUDI Name of team / customer, 4 characters
- ADA Type of model, 3 characters (for further types of models see legend mind map Figure 6.8 or list of abbreviations)
- ___ Three underscores for division between type and denomination of part or assembly, blank = underscore
- ___ Minimum of three underscores for division between denomination and abbreviation of designers name
- ABC Abbreviation of designer's name. First character = first letter of first name. Next two characters are the first two letters of the family name.

A.4.1.3 Definition of a new Version

Every designer solely works on its own file "Current". The file "Old" is only used for archival storage of old versions or variants. As soon as a new version has to be defined the part is opened in a new window to save the old version with the date of the day and "Save As" ("Save as new Document") in file "Old":
KOB_AUDI_KIN_001___BENENNUNG_DES_BAUTEILS____ABC__20091030
In this file the denomination of old versions or variants exactly has 60 characters.

A.4.1.4 Use of Adapters

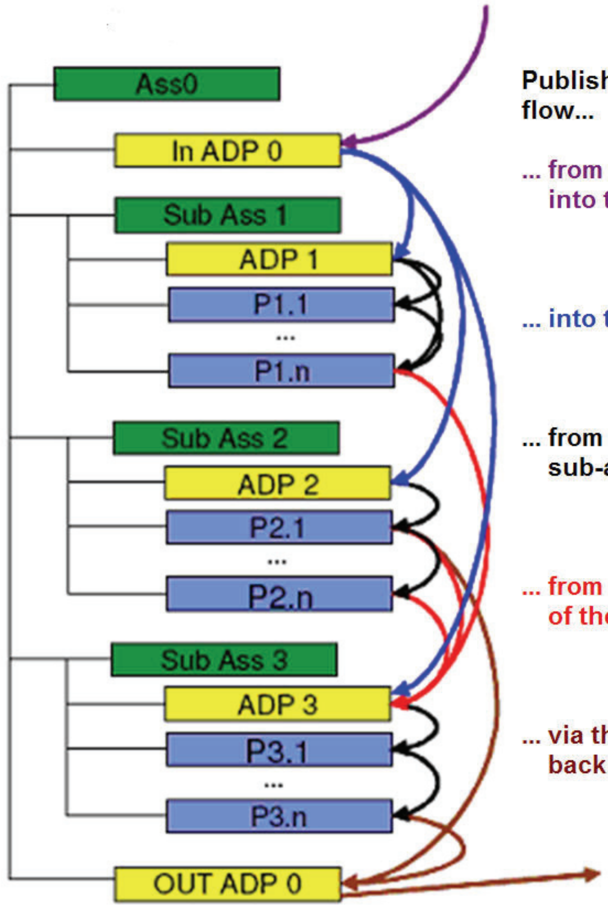


Fig. A.4.1 Hierarchical information flow in Adapter-files (Untiedt, 2009)

To uncouple the information flow within a CAD-model a hierarchical link-management is necessary (see Chapter 3.4). Every assembly-level therefore has one or more adapter files for the management of references (see figure A.4.1). Published references are copied with link into the lower hierarchical level (View-link). From the linked references isolated copies are generated which are the basis for the design

(Manual- link). For a clear arrangement internal references of the designer and external references are administrated in different files.

A.4.1.5 Preparation FEM-Calculation

Every designer must prepare his structural parts for FEM-calculation. In the start model a Geometrical Set (file) is defined where mean (mid) surfaces (controlled by the sheet thickness parameter) as well as joining elements are stored with defined denominations for function and position in single sub-files under support of the feature “Copy, Paste special with Link”.

A.4.1.6 Denomination of geometrical Elements within CAT Parts

The name of a Geometrical Set (file) and its last element always have the same name. Denominations of published elements at first contain information about the type of geometry: PT Point, LN Line, CV Curve, PN Plane, SF Surface, and PM Parameter.

A.4.1.7 Publications

Links are exclusively defined with published elements. Published elements are never cancelled, otherwise links become orphans (parentless). All elements provided for publication are stored in the Geometrical set (file) “To_Publish”. In this file the elements get their denominations and are “packed” (converted) with the following functions: Points become “Affinity”-elements, Lines, Curves, Planes and Surfaces are converted with the “Invert”-feature. This is necessary for an easy “Replace” or “Synchronisation” of data within an update.

A.4.1.8 Curve and Surface Directions

Curve directions are defined on base of the main axes system. The true most mapping of a curve on a plane of the main axes system defines the axis which defines the direction of the curve. Closed curves are defined counter clockwise. As

Class-A surfaces are visible surfaces their direction vector always shows from the visible surface into the part. For all other part surfaces of sheet metal parts always the male mould surfaces are designed (inner side of the deep drawn part) and the direction vector shows into material thickness (towards the outer side of the deep drawn part).

A.4.1.9 Multi-Model-Links (MML)

Only Reference-to-Instance links (CATIA Import-links, KBE-links and Context-link) are used within the project. This means that both the file with the reference geometry (transmitter) and the file where the new link is stored (receiver) must be opened in the same window while the reference is copied and pasted to define the link.

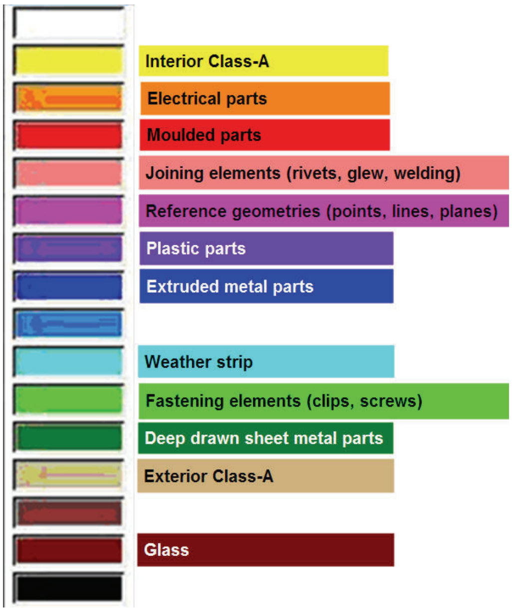


Fig. A.4.2 CATIA-standard colours to identify certain elements / surfaces

A.4.1.10 Use of Templates

It is the target of the project to use small templates to reduce the efforts for recurrent design work. The templates should be defined in a way which allows their easy application. Templates are defined by single designers of the two teams approved by one of the team leaders and provided in the project file for all other members. For every template under support of the “URL & Commands” feature brief instruction are provided stored in the project file as well.

A.4.1.11 Colouring of CAD Geometries

CATIA-standard colours are used to distinguish CAD-geometries (see Figure A.4.2)

A.4.1.12 Design Parameters

For the common use within the projects carried out for AUDI and GFI (used as an example in chapter 6) some design parameters were defined at the beginning:
Joining technique punch riveting:
Minimum width of flanges: 16mm, rivet direction always from thin to thick metal sheet, maximum distance between two rivets: 60mm, the accessibility for specified rivet guns must be assured.
The design must be built in a way that allows all optional sheet metal thicknesses between 1…2.5mm. Applicable sheet thicknesses (t) are: 1.0; 1.1; 1.2; 1.3; 1.5; 1.7; 1.8; 2.0; 2.2; 2.4; 2.5.
The minimum fillet radii are 2.5 to 3.0 * sheet thickness.
The release angle for deep drawn aluminium sheets depending on the depth of the drawing has a minimum value of 7°.
The minimum distances between interior Class-A surfaces and sheet metal parts is 15mm, for supporting glue between structure and exterior panels 4…5mm.

A.4.2 Start Model

At the beginning of Parametric Associative Design (PAD) in automotive body engineering and still all around CAD-models are structured not at all or very individual which makes the models illegible for third people. It is necessary within an environment of designing groups to define regulations for the systematic proceedings for design, development and collection of geometries and information and for administration in the hierarchical structural tree of a CAD-model.

In the first projects for the automotive industry the designers of the student teams could individually define their own methodologies and structures. They were only asked to use the hierarchical principle of IDO (Input, Design and Output). In the meantime the author experimented a lot with the structural tree and the design proceedings combined with the structural tree (compare with Chapters 3 and 4).

Except OPEL all OEMs located in Germany use the CAD-System CATIA. The CATIA work group AK 4.6 (CAD/CAM) of the German automotive industry has elaborated and issued a common regulation in 2009 (AK 4.6, 2009). A common CATIA V5 start model was verified in an OEM comprehensive work group with focus on CAD/ CAM strategy. This start part is based on verified design methods and intends a standardisation of CAD requirements and therefore a facilitation of CAD performances by engineering partners. Guidelines are established, documentations are developed and example models are generated. The OEM start model was validated in several passes together with engineering partners. Since May 2009, it is used in pilot approaches in AUDI.

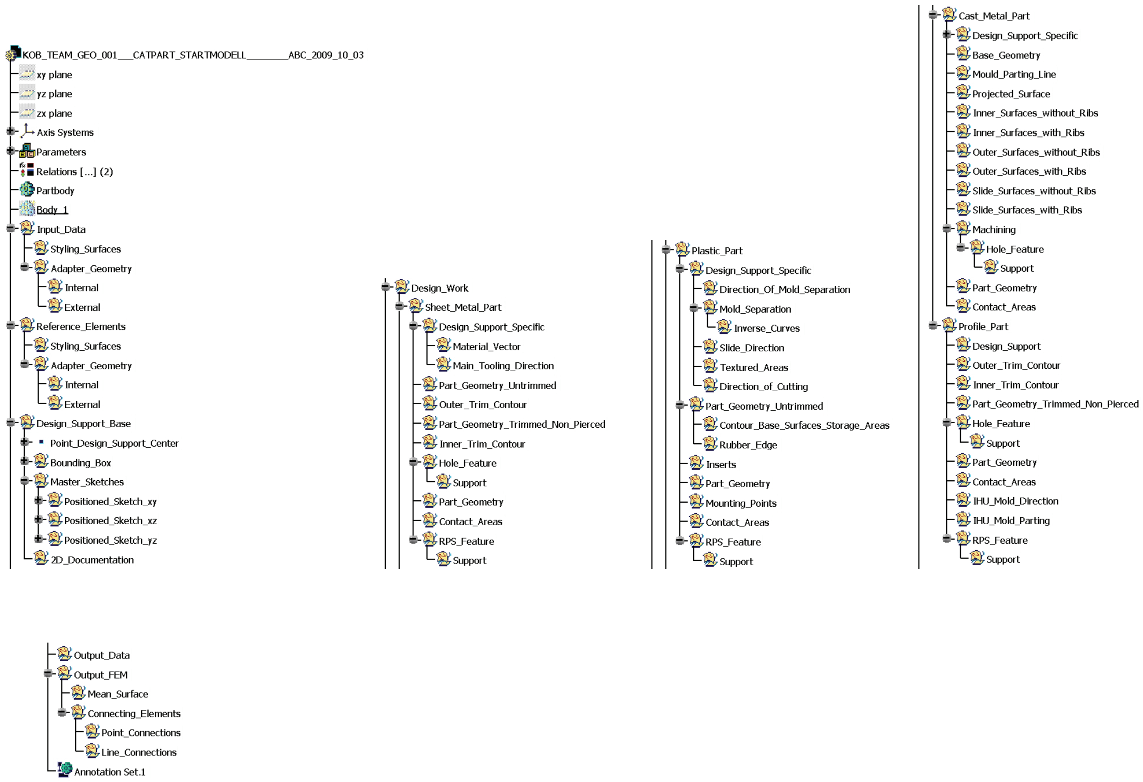
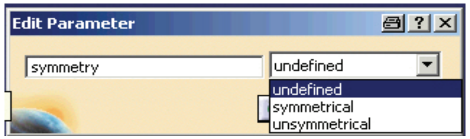
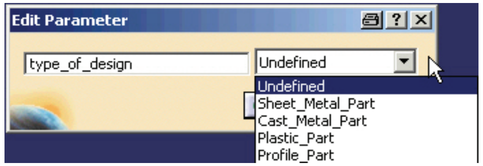
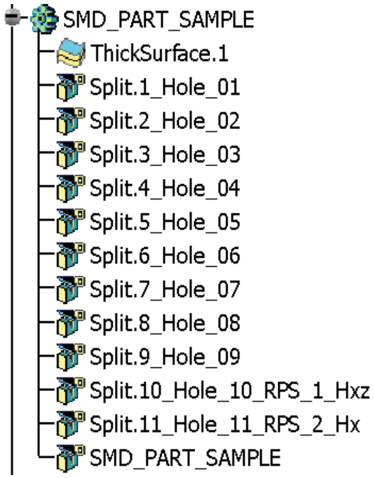
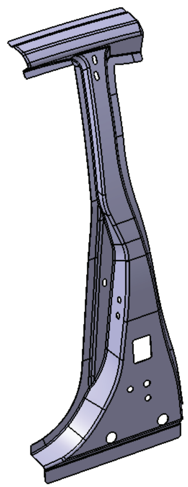
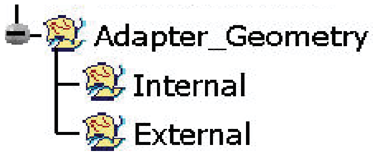
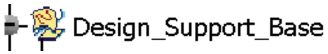


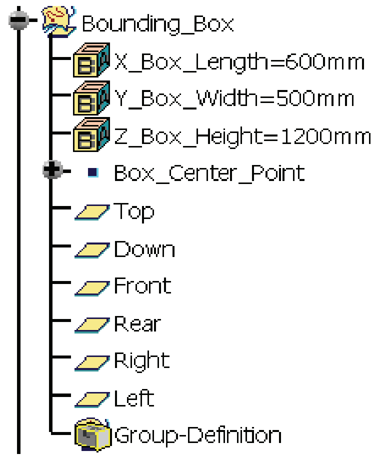
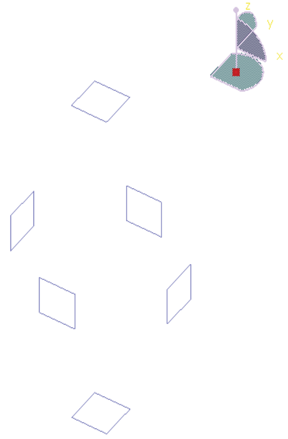
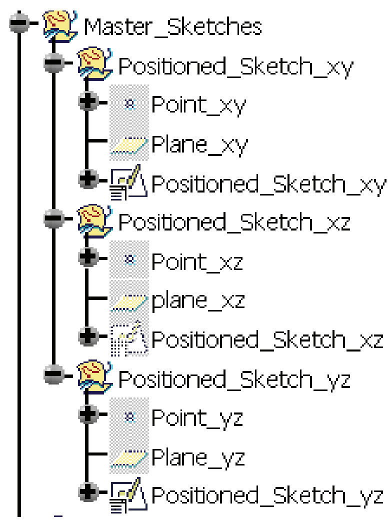
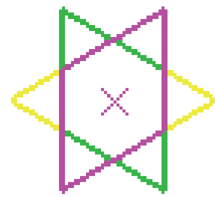
Fig. A.4.3 Start model structure of teams AUDI and GFI acc. to Bode et al, 2010 and Untiedt et al, 2010

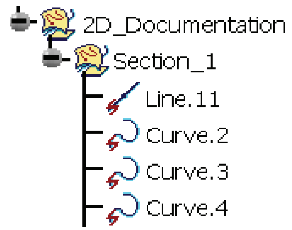
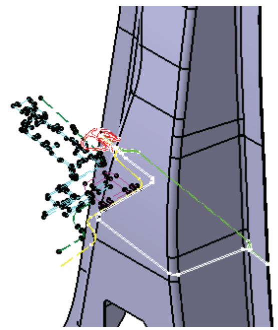
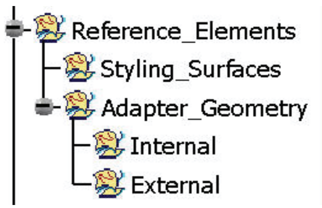
Since 2009 the author uses a reduced OEM start model in his lectures in semester 2 and 4 of the Bachelor programme. For the project teams AUDI and GFI the version tested by AUDI was adjusted for the necessities of the project work in the concept phase. The start part contains a common area plus special areas in its structural tree for different manufacturing techniques such as deep drawing, tailored blanks, casting, plastic injection moulding, forging, extrusion (profiles) / extruding and hydro forming. Figure A.4.3 shows the structure of the start part and Table A.4.1 describes the content of the start model on the example of deep drawn sheet metal parts in detail:

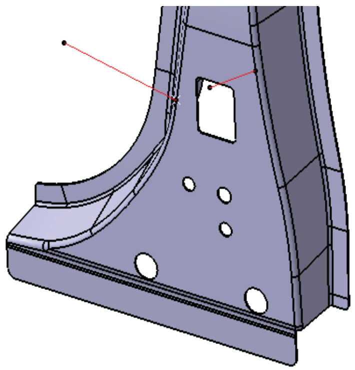
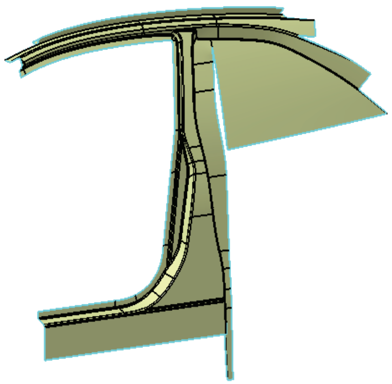
Geometrical Set structure/ Element name	Example structural tree	Example geometrical element	Explanatory notes
Parameters	Parameters	%	Parameter set with individually predefined parameters.
material=	material=	22MB5Ti / H340	For documentation purposes material denomination should be added
material_density=	material_density=0kg_m3	%	Density of material necessary for calculation of weight
material_thickness=	material_thickness=0mm	%	Parameter of constant thickness of sheet metal part
weight=	\`weight\` =0kg=\`volume_solid\` *\` ma		Calculation of weight by density and volume of Part Body
volume_solid=	\`volume_solid\` =0m3=smartVolume(		Volume of Part Body
symmetry=	symmetry=symmetrical	symmetrical	Defines the symmetry properties of the Left Hand Side (LHS) part. The sheet metal part is "symmetrical" as long it is symmetrical to XZ-plane and has symmetrical part on the Right Hand Side (RHS).
undefined symmetrical unsymmetrical			undefined --> Part (Part No.) only appears once, is symmetrical to XZ-plane, or appears several times. symmetrical --> LHS part and RHS parts are symmetrical unsymmetrical --> LHS Part (Part No. for LHS part) and RHS Part (Part No. for RHS part) are different.
type_of_design=	type_of_design=Sheet_Metal_Part		Defines the part-type. In this case: SMD_Part, i.e. Sheet Metal Design

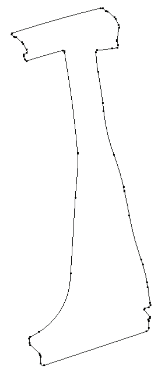
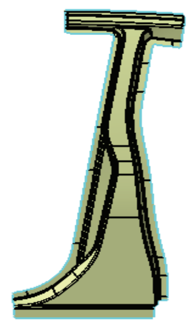
Geometrical Set structure/ Element name	Example structural tree	Example geometrical element	Explanatory notes
undefined Sheet_Metal_Part Cast_Metal_Part Plastic_Part Profile_Part			Part types available in start model
Part_Body			Part Body, Final part (Solid), Name arbitrary. Final Part must be completed as measurement of volume and calculations of weight require the input. (For sheet metal parts in most cases the "Thick surface" feature is used to generate the solid. The surfaces of set Part_Geometry_Trimmed_Non_Pierced should be used for this design operation. Holes are only designed perpendicular to sheet surfaces on planar parts. All other holes are designed in tooling direction under support of auxiliary surfaces (e.g. extrudes) and the feature "Split" for surface models or Boolean Operation "Remove" for solid parts. It is NOT allow-ed to use the Part design feature "Hole" as this feature implies the use of BReps.
Input_Data			(Adapter) Contains references from external sources. All references are inserted with active link to the original document. All references of this file are NOT used for design and only used for View-Link purposes. References used for design are the isolated copies placed in the set "Reference_Elements".

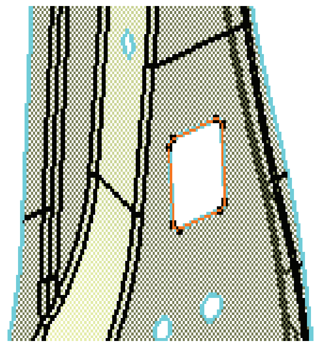
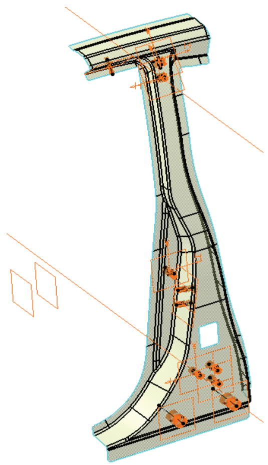
Geometrical Set structure/ Element name	Example structural tree	Example geometrical element	Explanatory notes
Styling_Surfaces			(Adapter) Contains Class-A surfaces from Adapter models on higher hierarchical levels (see Input_Data)
Adapter_Geometry			(Adapter) references from Adapter models on higher hierarchical levels (see Input_Data) In "Internal" references from product-levels of the assembly or subassemblies are inserted. In "External" references of higher hierarchical product levels are inserted. This differentiation and further sub-files (Geometrical sets) improve clearness and traceability.
Design_Support_Base			For all areas available predefined auxiliary geometries
Point_Design_Support _Center			Origin of auxiliary geometries

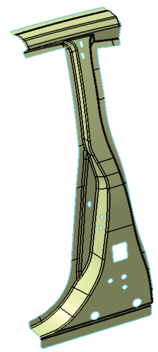
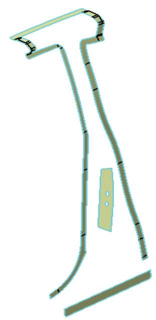
Geometrical Set structure/ Element name	Example structural tree	Example geometrical element	Explanatory notes
Bounding_Box			In this set planes are defined which can be used to border the designed space. They are mainly used for the "Near-Operator" when one result of multiple design results has to be clearly chosen. Planes of the bounding box can be used within the Sketcher or for trimming of large reference geometries. The "Box_Center_Point" which depends on "Point_Design_Support_Center" can be moved, positions of planes can be defined individually. Normally one Bounding Box is defined around the designed part. In some cases it is necessary to copy the Bounding Box for a second or more design areas.
Master_Sketches			In this set Master Sketches (concept sections) are defined. As long as the Master Sketches have to be defined perpendicular to 3D curve, point, plane and local axes system have to be designed individually. According to "Point_Design_Support_Center" three positioned Sketches are predefined parallel to the planes of the main axes-system. ONLY Positioned Sketches are used which can be easily positioned within the design space.

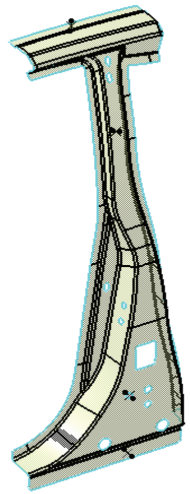
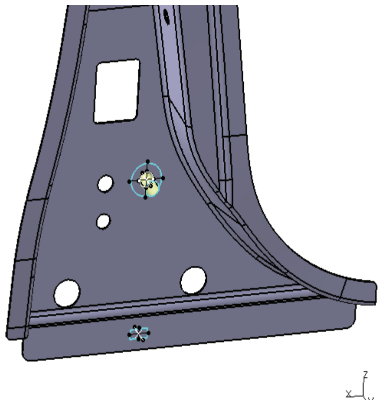
Geometrical Set structure/ Element name	**Example structural tree**	**Example geometrical element**	**Explanatory notes**
2D_Dokumentatitio n			This set stores isolated environment-geometries (e.g. sections) used for drawings or principle sections.
Internal_Reference_ Elements			In this set the isolated copies (Manual Links) of the reference geometries (View Links) are administrated. This set is an exact image of the Input_Data Set, but all references are isolated. The references of this set are used for the design. For a stable synchronization all reference geometries should be "Invert-Features", points "Affinity-Features".
Design_Work			Underneath this set the design takes place. For several different manufacturing methods sub-sets for specific contents are defined.
Sheet_Metal_Part			Underneath this set the order of primary and secondary sub-surfaces of a sheet metal part is designed.
Design_Support_ Spec.			Specific supporting geometries

Geometrical Set structure/ Element name	Example structural tree	Example geometrical element	Explanatory notes
Geometrical Sets Material_Vector and Main Tooling_ Direction	Material_Vector Point.18 Line.301_Material_Vector Material_Vector Main_Tooling_Direction Plane.54 Inverse.35 Project.4 Line.302_Main_Tooling_Direction Main_Tooling_Direction		This set must be completed. The material vector shows the direction in which material is applied to the designed surface. In all wire frame and surface models this vector is essential. The length of the vector (line) is the hundredfold of the sheet thickness. The main tooling direction must be defined at the beginning of concept design. Even when part size is undefined a tooling direction can be defined on the basis of two points and a line or designed under support of the shape of the Class-A surface. As long as the part design depends on a tooling direction the two points for the defined direction can easily be optimized and part surfaces can be updated.
Part_Geometry_ Untrimmed	Part_Geometry_Untrimmed		In this set untrimmed primary surfaces are stored.
Individual Geometrical Sets for the design of wire frame and surface geometries (Name of GeoSet and last feature must be identical)	Base_Surface Front_Flange_Untrimmed Rear_Flange_Untrimmed Group-Top_Area_Surface Top_Area_Surface Group-Case_Front Case_Front Case_Back Assembly_Of_Part_Untrimmed Part_Geometry_Untrimmed		In an individual structure all basic primary and secondary surfaces are designed and ordered according to their development hierarchy. The hierarchical assembly of the primary and secondary surfaces is carried out with "Fillet" and "Trim" operations. As soon as sub-hierarchies get too big, groups can be defined to shorten the current structural tree. At the end all surfaces combined by fillet and trim operations are combined to one "Inverse"-element and stored in the set "Surface Part_Geometry_Untrimmed".
Outer_Trim_	Outer_Trim_Contour		Outside trimming geometries.

Geometrical Set structure/ Element name	Example structural tree	Example geometrical element	Explanatory notes
Contour			
Individual Geometrical Sets for the design of trimming geometries (Name of GeoSet and last feature must be identical)	Outer_Trim_Surface Outer_Trim_Contour		In this sets geometries are defined for the trimming of the sheet metal part "Surface_Part_Geometry_Untrimmed". The trimming geometries are 3D contours which define extruded trimming surfaces according to tooling operation or a trimming operation is defined perpendicular to tooling operation under support of a 2D contour. In that case the 3D contour is the result of operation. As soon as more than one trimming operation is necessary two or more contours may be the result, called "Outer_Trim_Contour_n..."
Part_Geometry_ Trimmed_Non_ Pierced	Part_Geometry_Trimmed_Non_Pierced		Final part surface without holes.
	Split.40 Part_Geometry_Trimmed_Non_Pierced		In this case the "Part_Geometry_Untrimmed" was split under support of the "Outer_Trim_Contour". The result is converted into an "Invert-element" named "Part_Geometry_Trimmed_Non_Pierced".
Inner_Trim_Contour	Inner_Trim_Contour		Inside trim contours (not holes)

Geometrical Set structure/ Element name	Example structural tree	Example geometrical element	Explanatory notes
Individual Geometrical Sets for the design of trimming geometries (Name of GeoSet and last feature must be identical)	Sketches Cutout_Electrical_Wires Inner_Trim_Contour_Kabeldurchgang_Tuerstecker		According to the outside trimming contours one or more trimming contours may be necessary e.g. for cable grommets, steering column etc. The contours are defined in positioned sketches on planes perpendicular to the tooling direction. The inner trimming contours are the result of "Trim"-operations with extruded surfaces generated from the planar contours. The results of more than one inner trimming contours are defined "Inner_Trim_Contour_n..."
Hole_Feature	Hole_Feature		In these sets the hole-design is administrated.
Support	Support		Auxiliary geometry necessary for the hole design.
Individual Geometrical Sets for the design of holes (Name of GeoSet and last feature must be identical)	Hole_01 Hole_02 Hole_03 Hole_04 Hole_05 Hole_06 Hole_07 Hole_08 Hole_09 Hole_10_RPS_1_Hxz Hole_11_RPS_2_Hx Holes_Assembly		For the design of holes for all kind of parts (surface and solid designs) only auxiliary geometry defined with the "Extrude-Feature" of work bench GSD is used. In set "Support" the hole centre, centre line and tolling plane for the stamping operation as well as the hole contour and the extruded surface are stored. The holes are not designed in these sets. To show which extruded surfaces belong to one stamping operation the surfaces of several holes are merged by a "Join" operation. The "Join" is not used for the trimming! (The "Hole"-feature of the Part Design work bench is NOT used --> see above "Part Body")

Geometrical Set structure/ Element name	Example structural tree	Example geometrical element	Explanatory notes
Part_Geometry	Part_Geometry Split.41 Part_Geometry		Part_Geometry is the final surface model. In this set and its sub sets the trimming operations regarding inner contours and holes takes place. In the example the result is Split.41. Part_Geometry is the result of an "Invert" operation of the resulting Split.41. It is NOT allowed to design extracts of the final solid.
Contact_Areas	Contact_Areas		Geometrical sets for the contact surfaces to mating parts
Individual Geometrical Sets for the design of contact surfaces (Name of GeoSet and last feature must be identical)	Extract_Lower_Joining_Area Extract_Lock_Claw_Area Contact_Area_Lock_Claw Contact_Area_Lower_Joining Top_Joining_Area_Extracts Contact_Area_Top_Joining		In this set contact surfaces to mating parts are designed and collected. If possible intermediate results of the part surface design should be used for this collection. In some cases "Extract" operations must be made to get the contact surfaces expected. At the end of every contact surface design an "Invert"-operation is made. If necessary several corresponding contact surfaces are combined by a "JOIN" operation.
RPS_Feature	RPS_Feature		RPS (Reference Point System)-geometry for manufacturing purposes.

Geometrical Set structure/ Element name	Example structural tree	Example geometrical element	Explanatory notes
Support	Support Plane.57 Plane.59 Plane.60 Intersect.39 Plane.61 Intersect.40 Plane.63 Intersect.43 RPS_3_Fy Intersect.53 Line.305_Axis_3_Fy RPS_4_Fy RPS_5_Fy RPS_6_fz		RPS is a system of dimensional accuracy points in a part or a welding assembly used for a clearly defined position in design space, for fixation on mounting or measurement devices. The auxiliary geometries necessary for the definition of the RPS system are designed in the support set. It is useful to use sub-sets to keep the design traceable. The supporting geometries for RPS_1_Hxz and RPS_2_Hx of this example have already been design under the "Hole"-features. RPS-points, RPS-hole centre lines.
RPS_01_Xnn and the following	RPS_01_Hxz RPS_01_Hxz_Holedirection RPS_01_Hxz_Point RPS_01_Hxz_Boundary RPS_02_Hx RPS_03_Fy RPS_04_Fy RPS_05_Fy RPS_06_fz		Geometrical Set with all RPS geometries relevant for the sheet metal part. The denomination of the sets has an ascending order. The fixing directions are defined by the names of the geometrical sets. All geometries must be designed as "Affinity" features for points and "Inverse" features for axes, surfaces, planes etc. RPS-point, In SHOW RPS-Hole direction --> [3D line in RPS point with length of 10x local sheet thickness and defined direction of material flow from RPS point to offset direction], In SHOW RPS-Boundaries, In HIDE RPS-Vector --> [Surface normal, 3D line in RPS point

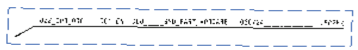
Geometrical Set structure/ Element name	Example structural tree	Example geometrical element	Explanatory notes
			with length of 10x local sheet thickness and defined direction of material flow from RPS point to offset direction], In SHOW RPS-Surface or Plane, In HIDE
Output_Data	Output_Data		In this set all geometries (if possible "Invert"-geometries) are collected which are provided for transfer to other CAD-models. The output geometries should be structured under support of sub-sets according to Input_Data and Reference_Elements sets.
Output_FEM	Output_FEM Mean_Surface Connecting_Elements Point_Connections Line_Connections		References for FEM calculations In set "Mean_Surface" the middle surface of the part (Offset controlled by 0.5 x sheet thickness) is stored. Connecting geometries are structured according to the joining techniques used. ▪ Point_Connections" are riveting or spot welding points ▪ Line_Connections" are gluing or laser welding joins
Annotation_Set.1	Annotation Set.1 Views Notes		In this set all denominations, explanatory notes, markings and dimensions defined by the features of the Functional Tolerancing & Annotations (FTA) work bench are automatically stored. All elements are defined to improve documentation and traceability of the CAD-model.

Table A.4.1 Detailed description of start model (Example Sheet Metal Part) acc. to Bode et al, 2010 and Untiedt

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