



THESIS FOR THE DEGREE OF DOCTOR OF PHILOSOPHY IN PRODUCT AND  
PRODUCTION DEVELOPMENT

# CAD-MODEL PARSING FOR AUTOMATED DESIGN AND DESIGN EVALUATION

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# ABSTRACT

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Product development has both innovative and analytic sides. Starting from the requirements, a design suggestion is generated. In order to assess how well the envisioned design fulfils the requirements, it is sometimes necessary to build a computer model of it for the analysis. The overall motivation of the work presented is to reduce the time spent on creating the model by reusing knowledge gained from developing similar products by suggesting, building and evaluating IT-systems. To verify the systems real design examples, obtained from companies that have participated in the research projects have been used.

The work is based on two major application examples. The first, involving the automated geometrical idealisation of die-cast parts (Paper I-III), and the second involving manufacturability of powder metallurgy pressed and sintered parts (Paper IV-VI). The work starts from the point in the product development process where it exists a design suggestion represented as an arbitrary format CAD-model. In the powder metallurgy case the object is to secure that the geometry is suitable for the production process. In the die-casting case the object is to automatically create an idealised version of the model for shell elements meshing. These two tasks have previously been treated as two separate cases, addressed by completely different software. This thesis suggests a common method for addressing the two cases. The method is based on converting the CAD-models, using the geometrical restrictions of the production processes, into a format with a specialised feature structure, parameterisation and construction history using a feature recognition approach. The features are then automatically reconstructed in a target CAD-system. The resulting, specialised CAD-model can be used for automated design and design evaluation purposes, demonstrated in the thesis. The models are therefore called DAR (Design Automation Ready)-models. The DAR-models are useful in that they separate the conversion from the subsequent treatment of the models providing modularisation, flexibility and user insight in the model structure. In that a construction history and parameterisation have been constructed in the target CAD-system, the advanced geometry manipulation and means for knowledge management often provided in modern CAD-systems can be accessed in a transparent and user manageable way. This extends the usefulness of the CAD-systems from involving only interactive work to managing all components sharing the same production process.

**Keywords:** Design automation, CAD, KBE, Feature based modelling, Feature recognition, Product development.



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The work has been conducted in two major research projects in close cooperation with a number of Swedish companies: Husqvarna AB, Höganäs AB, Callo Sintermetall AB, Volvo Powertrain AB and others. The incitement for starting to formulate of the research questions has to a large extent come from perceived problems in the daily activities of these companies. They have also been of great help in providing actual design problems for the verification of the systems and also helping to focus the work on real industrial problems. They are therefore greatly acknowledged.

I also wish to thank Professor Staffan Sunnersjö at the department of mechanical engineering in Jonkoping University. He is acknowledged for employing me at the department, starting the research projects that made this thesis possible and also for encouragement, mentorship and good advice when conducting the work. All colleagues at the department also deserve special thanks for interesting and fruitful discussions on the inspiring subject of IT support for the product development process.

Finally, I wish to thank my wife Katarina and my two daughters Klara and Julia for putting up with their husband and father constantly having IT systems and product development on his mind.



Roland Stolt

Jönköping, September 2008



# APPENDED PAPERS

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This thesis is based on the six papers (I-VI) listed below. Each paper is followed by a short summary.

## Paper I

Stolt, R. and Sunnersjö, S. (2005), "Automatic Preparation of CAD-generated Solid Geometry for FE-meshing", Proceedings of NAFEMS World Congress 2005, May 17-21 St. Julians, Malta.

Starting from a 3D CAD model of a die-cast part, this paper describes how automated routines, programmed in a CAD-system, can be used to construct a mid-surface idealization from the solid model. The purpose of the mid-surface is to provide a target for shell element meshing. The systems process the features as they are added. The method is based on the geometry being oriented in the tooling draft direction, a requirement for the production process to work. The motivation is saving time by automatically reusing a known manner of target surface creation.

## PAPER II

Stolt, R. (2005), "A CAD-Integrated System for Automated Idealization of CAD-Models for Finite Element Analysis", Proceedings of DETC2005: ASME International Design Engineering Technical Conferences and Computers and Information in Engineering Conferences -31st Design Automation Conference, September 24-28 Long Beach, CA, United States.

The paper recognizes that the cast geometries dealt with in Paper I often are of mixed dimensionality, requiring the partition into thick and thin regions. In order to handle this, it is proposed, and tested in practice, to identify company specific features by the dimensions of the geometric elements in their sketches. Proven representations can then be inserted in the FE-model to account for the effect of the represented solid geometry regions. A number of such product specific features are identified, and it is shown how the system can identify them and insert the correct representations and integrate them with the rest of the idealization. The time saved in the process is also estimated.

## Paper III

Stolt, R. (2006), "Reusing CAD-models of Die-cast Products for FEA", Proceedings of 2nd NAFEMS Seminar: Prediction and Modelling of failure using FEA, May 31 - June 1 Copenhagen / Roskilde, Denmark.

To estimate how well commercial FE pre-processors perform when automatically creating a target surface for shell element meshing from thin-walled geometries, a number of complicated parts that are currently in production are tested in two commercial pre-processors. The result is that individual surface patches are found, but the trimming of them to form a connected set of surfaces is not completed on the tested samples in either of the pre-processors. The method of identifying features developed in Papers I and II is extended to include neutral format CAD-models in a user interaction approach.

#### **Paper IV**

Stolt, R. (2008), "Identifying Features in CAD-Models for Powder Metallurgy Component Evaluation.", Proceedings of Design 2008: The 10th International design conference, May 19-22, Dubrovnik, Croatia.

The identification of features in arbitrary format CAD-model design suggestions is extended to the powder metallurgical pressing and sintering process, requiring a new set of features. The identified features in an arbitrary CAD-model are reconstructed in a parameterized version using the geometrical shape restrictions of the process. The reconstructed model largely facilitates the rule-based evaluation of the parts manufacturability when compared with basic rules and recommendations of the process, which is shown in the paper.

#### **Paper V**

Stolt, R. (2008), "An Automated Design and Evaluation System for Pressed and Sintered Powder Metallurgy Parts", Proceedings of MPIF/APMI 2008 World Congress on Powder Metallurgy and Particulate Materials, June 8-12 Washington D.C, United States.

In addition to the feature recognition and manufacturability evaluation process elaborated in Paper IV, the relationship between the part requirements, the pressing tooling and the geometry of the part is discussed. It is concluded that the part's geometry cannot be established and evaluated with any precision if the function of the pressing tooling is unknown. The possibility of creating a system for automated tooling design corresponding to the shape of the part and the mechanical requirements is discussed in the paper, and a prototype system for automated tooling design is presented.

#### **Paper VI**

Stolt, R. (2009), "Managing Product and Process Knowledge in CAD-Systems", Accepted for publication in proceedings of NordPLM'09: The 2nd Nordic Conference on Product Lifecycle Management, January 28-29 Goteborg, Sweden.

The product and process knowledge used in the systems described in Papers I-V needs to be maintained and refined to reflect the development of both products and processes. The paper elaborates on what types of knowledge that needs to be managed and how the knowledge can be represented in a transparent way, allowing the user to take active part in the knowledge management process. It is also shown how the interactive approach presented in paper III for finding features in arbitrary die-cast CAD-models can be automated.

# ABBREVIATIONS

---

AI	Artificial intelligence
API	Application programmable interface
CAD	Computer aided design
CAx	Computer aided x (x is replaced with aided activity e.g. M (Manufacturing)).
CE	Concurrent engineering
CSG	Constructive solids geometry
CI	Computational intelligence
GUI	Graphical user interface
DA	Design automation
DAR	Design automation ready (Author's abbreviation)
DFA	Design for assembly
DFM	Design for manufacturing
DRM	Design research methodology
DSM	Dependency structure matrix
ES	Expert system
FEA	Finite element analysis
FR	Feature recognition
IGES	Initial graphics exchange specification
KBE	Knowledge based engineering
KBES	Knowledge based engineering system
KBS	Knowledge based system
KM	Knowledge management
MAT	Medial axis transform
PD	Product development
PDM	Product document management
PM	Powder metallurgy
STEP	Standard for the exchange of product model data
UML	Unified modelling language
VB	Visual basic
VBA	Visual basic for applications
VDA-FS	Verband der automobilindustrie - flächenschnittstelle





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## CHAPTER 1

# INTRODUCTION

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## CHAPTER OUTLINE

This chapter introduces the reader to the thesis and outlines the technologies and related work that will be further elaborated in the following chapters in the thesis. It presents the research questions and defines the scope of the thesis. Finally, the contents of the chapters are outlined.

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### 1.1 INTRODUCTION TO THE THESIS

Designing mechanical products is a question of finding a solution that can satisfy or surpass all functional and other requirements placed on the product and, at the same time, designing it so that it can be manufactured to as low cost as possible. It is therefore important to begin in the early stages of product development to carefully plan how it should be manufactured. Further, there is also a demand to keep the time spent on the development of the new product low in order to meet market expectations. Consequently, it is for most products no longer possible to first design the product considering only its function and appearance and then redesign it so it can be rationally manufactured. Integrating these two activities is the core idea behind design for manufacturing (DFM) see e.g. (Boothroyd, Dewhurst et al. 2002). To ensure that both the functional and the manufacturability requirements are met, cooperation between the design and the production departments from the earliest phases of the design process has been found to be highly important. This can be ensured by employing a concurrent engineering approach (Prasad 1996).

As a consequence of the above, computer support to aid the designers in the product design process should include not only functional aspects of the product but also the manufacturing of the product to secure that the final design can be economically manufactured. The computer support for product development (PD) which this thesis is focused on is computer programs that aid the designers in the development of new products. They do so by representing product and process knowledge from the previous development of similar products and reusing it with a degree of automation when designing the new product. This is done by codifying the product and process knowledge in computer systems. This assumes that the process of designing the product is well understood so it can be formalised into code. The role of the system is then to secure that important lessons learned from previous projects are not overlooked, improving the quality of the new design. Another motivation is shortening the time spent in the product development process. Time is thus freed for the designers so that they can spend more time on the innovative side of the PD process instead of being preoccupied in repetitive design tasks.

The knowledge can be codified in different ways. Rather than intertwining the knowledge in the code as is often done in regular programming, other programming techniques exist that allow the separation of the knowledge from the execution of it. The advantage gained from doing so is that managing the knowledge content of the program does not require that the code itself be changed. This would normally require programming skills and a detailed insight

into the structure of the code. The user can declare rules and facts, and then the program can determine when and how it should be applied. A code structure that supports the active knowledge management (KM) is fundamental in knowledge-based systems (KBS). KBS has roots in computer science and research in artificial intelligence (AI) and has found use, among a multitude of applications, in engineering. This is described in numerous books such as (Krishnamoorthy and Rajeev 1996; Sriram 1997; Hopgood 2001). Another commonly used term is expert system (ES). An ES is a KBS applied in a specific domain of knowledge.

To define the geometry of a product, various CAD-systems are often used. Formerly the CAD-systems were merely aimed at the interactive definition of the geometry of the parts. However, lately they have come to include more tools for design synthesis and analysis enabling them to support a larger scope of activities in PD. The CAD-systems have also started to integrate functionality for KM, providing an environment in which product and process knowledge can be represented and automatically reused. The term knowledge-based engineering (KBE) has started to be used to describe such systems. It was first used to denote a KBS used in the engineering context (i.e. an expert system). Lately, the term has been given a broader scope incorporating functions for automation and knowledge reuse integrated in the CAD systems (Hoisl, Shea et al. 2008). These CAD-systems allow the creation of rules and procedures for the automated design and evaluation of geometry. The functions of the CAD-programs can be controlled by application programs hosted by the CAD-system. This enables the creation of automated systems that autonomously or with limited user involvement can perform design and design evaluation activities. External programs can also be created to work in conjunction with the CAD-program. The result can be said to be a KBE-system for specific engineering tasks.

## 1.2 AUTOMATED PROCESSING OF CAD-MODELS

The preliminary geometry of a product design exists in many situations. It may possibly have been created by a prospect customer who wants to get a quotation on the production cost for a suggested design. Similarly, within single organizations, supplier costumer relationships can emerge, such as when the design suggestion made at the design department must comply with the needs of, for example the production department. This process typically starts with the designer making a preliminary design suggestion. This suggestion has to be discussed with other functions within the organisation to revise it to comply not only with the customer requirements on the part, but also with all the other aspects of the product's life cycle.

It is this evaluation of the design suggestion for various purposes that is in focus in this thesis. Using the KBE tools aforementioned, it should be possible to define systems to automatically check that these requirements have been taken into account, thereby avoiding that any aspects are overlooked. Over time the requirements for the products change requiring that the KBE tools provide an environment to support active KM. Consequently, one of the aims of the research presented here is to explore how such programming environments can be used to manage and reuse general product and process knowledge related to specific production processes to produce the suggested designs. This is explored through two major application examples. The first example, elaborated in Papers I-III, involves the creation of a surface representation of the thin-walled solid CAD-model geometry of nearly tooling-ready die-cast parts. The purpose is to automatically provide a target surface for shell element meshing so that a light weight shell element finite element analysis (FEA) model may be constructed from the solid model. This is motivated by saving time and raising the quality of the target surface, as opposed to creating it interactively in the CAD-system. The creation of the target surface can be seen as a repetitive design task involving product and process knowledge from previous

projects. The solid geometry CAD-model is automatically processed so that a best practice target surface can be constructed.

The second example, described in Papers IV and V, involves a tool for supporting the design of pressed and sintered powder metallurgy (PM) parts. Design suggestions for the parts are evaluated for their manufacturability using a rule-base including rules and recommendations on how to make the parts most suitable for the production process. It is primarily an advisory system that starting from a CAD-model containing the design suggestion can check that the geometry complies with the geometrical recommendations of the process. This facilitates the creation of a tooling for manufacturing the parts and secures its efficient operation in production.

### 1.3 PROGRAMMING CONTEXT

The two application examples both involve the automated processing of CAD-models. In the first case it is involved as starting point for an automated design task. In the second case, it is as an evaluation of a design suggestion. Both tasks involve engineering knowledge expected to have to be updated continuously, which must be reflected in the tool. Modern CAD systems provide, as mentioned, this type of functionality. However, the systems are primarily intended for interactive use on models that have been defined in the particular modeller. A procedure is needed that, starting from an arbitrary CAD-model, can convert it to a format that allows the full functionality of KM capabilities of the CAD-system to be used. An idea which emerged and was explored during the course of the projects was to use the geometrical restrictions of the parts imposed by the production processes. An automated procedure could then be established which formats the model in a way that can be interpreted by the target CAD-system. This is also the motivation for the title of the thesis “CAD-model parsing for automated design and design evaluation.” The CAD-model is said to be parsed, automatically interpreting its geometrical elements according to the indented production process and the following design and evaluation activates. The parsing of the CAD-models and the following treatment of it, require the use of engineering knowledge both general and specific to certain products. How this knowledge is managed and reused in the different engineering tasks has also been explored. The indented process is shown in Figure 1.1 below.

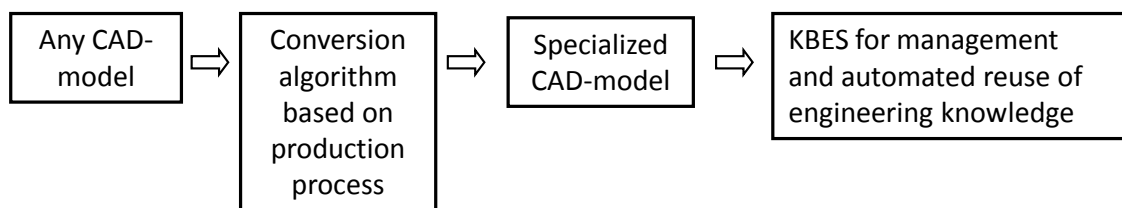


Figure 1.1 Formatting of CAD-model based on production process.

### 1.4 THE PRODUCT DEVELOPMENT PROCESS

PD is an iterative process. Its nature is both creative and analytic. The creative side involves trying to find a solution that can satisfy or surpass the requirements. This is sometimes referred to as design synthesis since the object is to synthesize the requirements into a single solution. The role of the analytical side is to check to what extent the requirements have been fulfilled by the suggested solution. In the Figure 1.2 below, adapted from (Johannesson, Persson et al. 2004) the interaction between the analytic and the synthesis side is explained.

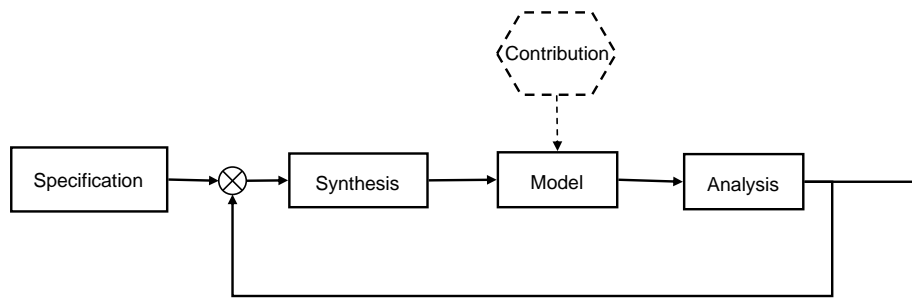


Figure 1.2 The PD process adapted from (Johannesson, Persson et al. 2004).

It begins with a requirements specification followed by design synthesis. In order to evaluate the suggested design, a model of it is needed. If the evaluation of the model shows that the solution can fulfil or surpass the requirements, is accepted for further development. If not, another loop is needed where an alternative solution is synthesized and evaluated. This process continues until a satisfactory solution has been found.

The contribution (refer to Figure 1.2) made in this thesis is in the model building activity and in particular in the reduction of the time required to transfer the design suggestion into a model which can be evaluated. This is mostly done by proposing and exploring automated or interactive support tools of accomplishing the model creation. This will shorten the time required to evaluate the candidate solution, shortening the time spent on the whole PD process.

## 1.5 RESEARCH QUESTIONS

The research questions have emerged during the course of the work and have come to centre around the production process related formatting procedure and how the CAD- systems can be used to manage the various types of knowledge needed to complete the tasks. A summary on how engineering knowledge is managed and reused in the two application examples is given in Paper VI. Research questions Q1-Q3 relate to both the production process based formatting procedure and the knowledge management and reuse issues. They are expressed below:

- Q1. How can a preliminary geometrical specification of a product together with production process constraints be used as a starting point for further automated design and evaluation of the product?
- Q2. How can the need of transparency and maintainability of product and process knowledge be addressed when exploring Q1?
- Q3. How can the above support the development of new products?

## 1.6 SCOPE OF THE THESIS

The research questions have to be seen as primarily aimed at studying how they apply to parts produced by the two manufacturing processes. Further, only single components are discussed in the thesis. The components are typically meant to be assembled into complete products, but design for assembly (DFA) issues are not discussed. The work has instead been focused on the manufacturability of the PM parts and, in the case of the die-cast geometries, structural analysis. However, the actual analysis is out of the scope of the thesis. The focus is on the

automated design of the target surface, including the knowledge of how to make a best practise representation of the part to suit the purpose of the FEA.

A number of representative components manufactured by the die-casting and PM pressing and sintering process have been studied, forming a starting point for reasoning around the research questions. There are limitations as to what type geometries can be processed in the experimental systems created to study the application examples. The systems should not be seen as a generally applicable tool, but rather as an exploration of the possibilities of extending the KBE functionality of the CAD-systems to manage and reuse the engineering knowledge needed in the application examples.

## 1.7 THESIS OUTLINE

In the next chapter, Chapter 2, the scientific method is presented. How the work was planned and conducted to address the research questions and how the results were verified is described. The frame of reference is given in Chapters 3 and 4. Chapter 3 presents the different technologies that were used to conduct the work and build the experimental systems. In Chapter 4 the related work is presented, placing the thesis in the context of previously conducted work. Chapter 5 describes the production process-based algorithmic procedure developed to format the CAD-models. It explains the use of these models and forms the results section of the thesis. Chapter 6 presents the two application examples made to die-casting and PM pressing and sintering and provide a number of benchmark examples to evaluate the systems. Finally, Chapter 7 discusses the conclusions that can be drawn from the results and answers the research questions. It also suggests how the work can proceed in the future.





## CHAPTER 2

# SCIENTIFIC APPROACH

## CHAPTER OUTLINE

This chapter serves to place this work within the scientific context. It also describes the scientific method used and explains how the results were verified and validated.

### 2.1 RESEARCH APPROACH

The research presented in this thesis is empirical and is based on hypothesis testing. This is a classical model for conducting research, as depicted in Figure 2.1 below (Roozenburg and Eekels 1995).

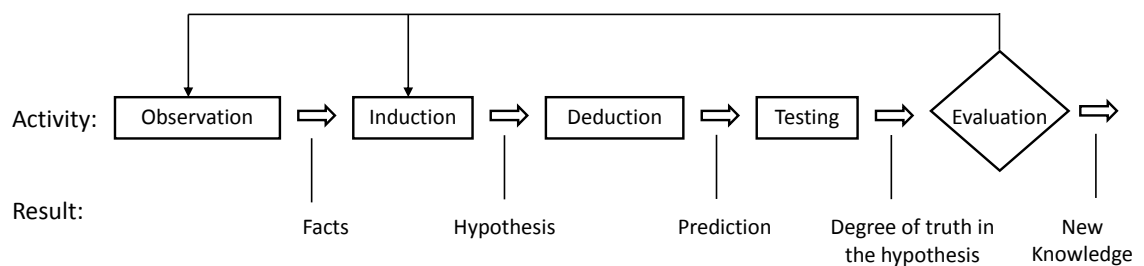


Figure 2.1. The empirical hypothesis testing research process as adapted from (Roozenburg and Eekels 1995).

The research process starts with observation from which the facts are gathered. The facts are generalized by induction into a hypothesis. Predictions are deduced from the hypothesis. These predictions are then tested to assess the degree of truth in the hypothesis. If nothing is found that is inconsistent with the predictions, the hypothesis is accepted as true for the time being. In this work the hypothesis is not explicitly expressed. Instead, three different and related research questions are presented (see Section 1.5). In order to investigate the questions, it was necessary to make assumptions regarding the answers to them. Otherwise, it would not have been possible to determine what the systems created for the answering of them should be focused on. The questions and the assumed answers together form the hypothesis, which is tested through the application examples.

#### 2.1.1 Artificial sciences

The classical science model is appropriate in natural sciences, aimed at providing explanatory models for phenomenon observed in nature. However, the research presented here is different in that it is aimed at the study and improvement of created things. This type of sciences is sometimes called artificial. They have the added purpose of inventing and improving rather than just explaining.

Examples of artificial sciences are business, architecture, computer science and medicine. This distinction between the artificial and the natural sciences has been discussed extensively by (Simon 1996).

Artificial phenomenon often rapidly changes over time since people strive for their constant improvements. This has a large impact on how artificial sciences can be conducted. Methods from natural sciences cannot be adopted directly. Instead pursuing artificial sciences show more similarity with the development process which was presented in the former chapter (refer to figure 1.2). Once the solution to a perceived problem have been synthetisized it is evaluated an if it is found to be an improvement over existing solutions it is accepted and further evaluated. All details on why it is better may perhaps not be explainable at the time.

(Wegenroth 2008) addresses the difference between science and engineering. Artefacts such as cars and aeroplanes cannot be designed following one coherent theory, still they have to be complete although perhaps not understood in all details. In the engineering case completeness takes precedence over truth, whereas science only deals with the quest for truth and does not have to consider completeness. In this thesis systems have to be designed and evaluated following the same procedure as in normal development with the difference that the intension is to addresses generally exciting problems common to a whole industry and not company specific problems. This will mean that general guidelines of how to support certain types of engineering work using suggested IT implements can be established. This is the primary goal of the research.

## 2.2 ENGINEERING DESIGN SCIENCE

This work presented in the thesis is part of engineering design science. It is clearly an artificial science, and its purpose is to study and improve the process of designing. This is mainly done by studying how actual design work is conducted and based on the studies give recommendations and propose tools and methods to support it. The field of design science can be divided into two main areas (Hubka and Eder 1996). First, there is the theoretical and practical knowledge about the designed artefact. The theory of the artefacts has, in reference to the discussion on completeness in the previous section, to be complemented with practical know-how and knowledge in order to complete the artefact. Second, design science seeks to enhance the actual process of designing. The design process also has to be complete. Therefore it, similarly to the artefact, also has a theoretical and a practical know-how side.

Putting the work presented in this thesis into this context one can say that is contributes to the knowledge about the design process. It seeks to formalize the design process so that it can be encoded in computer systems allowing it to be repeated. It also seeks to explore how the encoded knowledge can be kept up to date in an actual design process. The repetition must accommodate the variation in the designed artefact. The products share the same basic concept which is varied. The work also contributes to compiling and formatting knowledge about the designed artefacts so that it can be used in the proposed computer system. This knowledge mainly concern the production process shape restrictions of the PM and the die-casting processes.

### 2.2.1 Design research methodology

Since the aim is the development of new tools and methods to enhance the engineering design process, it is necessary to somehow judge how well the proposed tools and methods actually support the design process. This is needed in order to direct the development and to ensure that each contribution is comparable.

To address this, (Blessing, Chakrabarti et al. 2005) have proposed a design research methodology (DRM) aimed at providing structure and direction to the research process so that people involved in design research can better build on each other's contributions and to make the experimental work more focused and purposeful. The DRM is shown in the Figure 2.2 below.

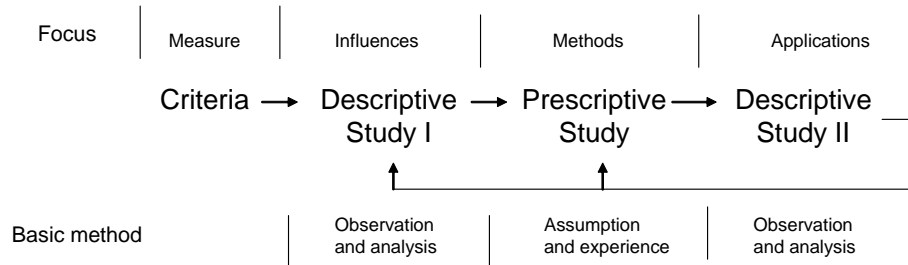


Figure 2.2. A design research methodology (DRM) adapted from (Blessing, Chakrabarti et al. 2005).

The research project should start by establishing measurable criteria. These criteria define the scope and focus of the project. It must be measurable in order to determine to what extent they are fulfilled. A study, called descriptive study I, follows. The object is to identify which factors can facilitate or prohibit the success of the intended method, providing guidance on what to focus on when developing a demonstrator in the following prescriptive study. This study will ensure that the demonstrator is purposeful and focused on the issues it was aimed to investigate. The prescriptive study is followed by another descriptive study evaluating the demonstrator to find how well the criteria were met, directing the research towards the development of precisely these functions. The DRM is iterative, often requiring a number of loops to refine both the actual programming of the demonstrator and the issues it is focused on (hence the arrows shown in figure 2.2).

The empirical science model, together with the DRM, has been used to structure the work behind this thesis. The interpretation of the empirical science model supported by DRM is shown in Figure 2.3. What in practice has been done and what purpose each activity fills is elaborated under each heading below.

Science model:	Observation	Induction	Deduction	Testing
Result:	Research questions	Assumed answers to the research questions	Predicted behavior of the system given the assumed answers	Verification of predictions.
DRM:	Criteria	Descriptive I	Prescriptive	Descriptive II
Activity:	Literature review Contacts	Planning of DA system	Programming of demonstrator	Evaluation of demonstrator

Figure 2.3. Activities in the research projects.

### 2.3 LITERATURE REVIEW AND CONTACTS

It is important to understand and build on results from previous research. Therefore a thorough review of previous and related work and technologies is required to begin with. The purpose of the investigation is foremost a knowledge build-up to grasp the work and

technologies related to the problems described in the two research projects. When systems to support certain aspects of the PD process are studied, this means finding what the difficulties in the development of previous demonstrators have been, and how well the intended functions have been fulfilled.

The main sources of information have been:

- Frequent contacts with practitioners and researchers.
- Review of textbooks, scientific articles and conference proceedings.

Some of the contacts with researches and practitioners have been made by visiting conferences and workshops. However, more important is the close corporation with companies that have participated in the research projects. Problems encountered at these companies have also made it possible to start formulating the research questions. The starting point for the formulation of the research questions is likely to be found in the daily practice of the engineers. The practitioners are well acquainted with the PD-processes in their companies. Consequently, they have the insight into how to discover problems or potential for development. If the problem encountered defies solution after consultation with expertise both in-house and externally then it is a good indication of the existence of a knowledge gap. However, the motivation for a research project can never be to solve a specific problem at a company. The role of the researcher is to place the problem in the context of previous research, judging if it is a generally existing unsolved problem. Note that it could be that something unknown is under investigation, so the direction a research project is going to take cannot be, in too much detail, planned from start. Decisions have to be taken on insightful guessing. If a research project is found to be motivated (i.e. it supports the knowledge build up at the company and is of interest to the research community), then the company's PD-process can provide an environment for field studies where observations can be made and experiments conducted. An important role of the participating companies is also in providing continuous feedback on the work conducted so that the most influential factors can be identified. Without this feedback, problems can otherwise get overly simplified and "academised". It may then be possible for the researcher to find sound scientific solutions to them. Unfortunately, if important factors have been overlooked, they are of no industrial use. This is because it was not the real problem that was studied. In this case, the found solution may in the best of cases, serve for educational purposes.

## 2.4 PLANNING THE EXPERIMENTS

When the research questions have been identified, a variety of methods exist to address them. Central in empirical research is that some form of experiment is devised to test the degree of truth in the predictions deduced from the hypothesis. In this case, the experiment involves prototype systems set up specifically to shed light on the research questions. Rather than formulating a hypothesis, the answers to the research questions have been assumed. This is really equivalent to forming a hypothesis through an induction process, since the assumed answers must be constructed so that they comply with all the observations made in the review. The questions and the assumed answers together form the hypothesis. Without a hypothesis it would not have been possible to devise the experiments to determine the degree of truth in it. It can be deduced that if the assumed answers to the research questions were true, then the system should display a predicted type of behaviour. This is consequently tested. The DRM emphasizes the importance of creating a focused system so that it will illuminate precisely the answers it was supposed to verify as much as possible. This requires careful planning before starting the programming of the system. In the case of proposing KBE systems, it is essential

to be clear about what type of design problems the system is intended to address. It should not be a limited selection of benchmark products. Rather, it should define a class of problems of general interest in engineering design. The type of design problems and the solution strategy must be completely clear. Often it is a matter of structuring and representing the problem in such a way that it can be formalized easily in the computer application.

## 2.5 PROGRAMMING

The creation of the demonstrator must be carefully carried out. If errors are present in it, then the influence of these can be wrongly interpreted. This could lead to erroneous conclusions being drawn from the experiment. The various components and the integration of them must be so well described that somebody else can repeat the same experiment and then be able to obtain the same results.

## 2.6 TESTING, VERIFICATION

The verification of the systems has been made by running a number of benchmark examples in them. These examples have been selected from in production parts, verifying that the systems function in the way predicted. The example parts are selected to represent the range of design problems that the system was intended to address.

## 2.7 THE RESEARCH PROCESS IN RELATED WORK

Nearly all references found in the literature review for this thesis were aimed at proposing demonstrator systems that can deal with a number of design and evaluation problems. The research questions that have been addressed in these publications have a broad scope, describing an intended use and proposing a complete system that can solve the intended tasks. Benchmark examples are sometimes provided, verifying the function of the systems. This reveals that the state of the research is still quite immature. Had there been a consensus within the scientific community as to what type or types of system/systems to use given the nature of the design task, then the publications would be directed towards working out the details of the these systems, rather than presenting complete new ones intended to demonstrate the feasibility of the proposed method. For the time being, the research seems to be in an explorative state, trying to grasp the big picture. It is possible that eventually a pattern will start to emerge from these explorations. The design tasks and solution strategies could be mapped together to form well motivated recommendations on how to create KBE systems to best support the various design tasks. This requires the research community to build on each other's results. If everyone starts fresh by redefining the problems then there will be no progress. Here contributions like DRM could prove helpful to the research community.

## 2.8 VALIDATION

When the results have been verified, they also need to be validated. In this context, the meaning of validation is to determine the usefulness and relevance of the results in industrial practice. This could be done by studying the impact on the design process. Do the results of the design process really improve per unit of spent resources? Designers could be put to work (aided by the systems), and reference groups could perform the same task using alternative methods in qualitative studies. However, given the immaturity of the technology, such studies are not yet possible. Further, if they were, methods and tools are constantly developing

so that when the results were available the methods and tools could already be obsolete. Instead, the systems are presented to practitioners working with the design of PM and die-cast parts. They were asked to try the systems and for their opinion as to how they think that the system will support the design process. Based on this input, the systems can be further developed until the users accept them and start using them as an integral part of the design process. This user acceptance is seen as a validation of the systems industrial usefulness.

## CHAPTER 3

# ENABLING TECHNOLOGY

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## CHAPTER OUTLINE

This chapter elaborates on the technologies that were used in the experimental systems. It focuses on programming, KM, and CAD issues. This elaboration is intended to enhance the reader's understanding of the related work, which will be presented in Chapter 4.

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### 3.1 FRAME OF REFERENCE

Chapter 3 and 4 form the frame of reference, which has two purposes. The current chapter seeks to provide the reader with an overview of the different technologies used in the experimental systems upon which the work in the thesis is based. Chapter 4 (Related work) describes previously made efforts within the same framework studying systems with the same or similar purposes. The reader is given an insight into the results of the related work so that the contribution made in this thesis can be positioned in the context of previous research.

As explained in the introductory chapter, the work presented here deals with design suggestions represented as geometric models. Therefore, CAD technology has a central position, as depicted in the centre of Figure 3.1 on the next page. The surrounding technologies support the creation of the automated systems to support the PD process. They also enable the design and design evaluation activities, as well as the managing of the engineering knowledge needed. To give the reader an elaborate view of these technologies, the basis of application programming and CAD-systems automation is described. DFM is central to this thesis. That is why the reader is introduced to the production processes, die-casting and PM pressing and sintering; to understand how the geometrical restrictions of the processes are related to the experimental systems presented. AI tools used to support this activity are also presented in order to reflect the necessity of keeping the engineering knowledge accessible and revisable to the user. Further, one objective of the systems is the creation of a mid-surface representation for shell element meshing. As a result, a section on geometric idealization and FEA is also been included to enhance the understanding of this application example.



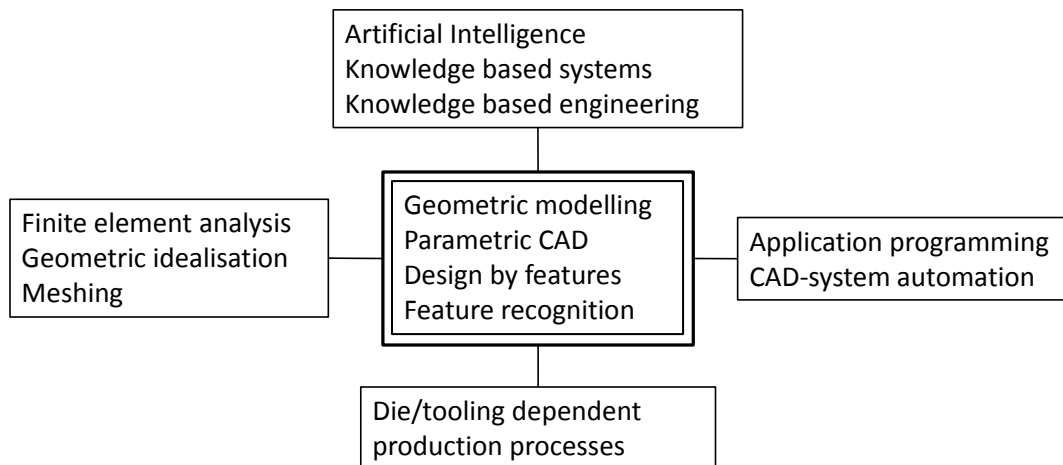


Figure 3.1. The different technologies used. The CAD-technology takes a central position.

### 3.2 GEOMETRIC MODELLING

Starting from digitized drawing boards, the CAD-systems have developed the capability to handle 3D geometries of single components and assemblies of several components. One important step in this development has been the introduction of solid-models. An elaborate view on geometric modelling is given, in, for example, (Mortenson 2006). From having to define all geometrical elements completely in 3D space, the models could now be defined by series of modifying operations such as constructive solids geometry (CSG). In CSG, the object is built up from geometric primitives through Boolean operations by adding and subtracting material. This leads to a hierarchical structure where the order in which the different objects are added is important. Consequently, the added and subtracted volumes are interrelated. If changes are made to a volume early in the series, the operations following will have to be updated accordingly. Later, rather than using geometrical primitives, the added and subtracted volumes were constructed from lower level entities by sweeping, lofting, and revolving operations. These increased the ease with which the modelling could be carried out (in particular making changes to the geometry). The different operations can be guided by sketches defined by the user. The sketch will then define a wireframe that is used to create a solid-body or a surface. The geometrical elements which are defined are assigned parameters such as the length of a line or the diameter of a cylinder. The parameters can be accessed and revised (hence, the term parametric CAD).

There can also be relationships between the added geometrical entities (i.e. constraints). The dependencies between the various geometrical elements are known as associativity. Making a change to one element will cause the dependant elements to update accordingly. The relationships are often visualized for the user in tree-like graphs. The graphs also show in which order the items were added to the model. These graphs are referred to as construction history trees. The items represented in the construction history tree are often termed features. Consequently, defining the geometry by adding a series of features to a model is called the design by features approach. It should be noted that the word “feature” has in the context of CAD a broader scope: it encompasses anything of interest in the model. Examples include tolerance and surface finishing features. Further, regions in the geometry that can be manufactured by different manufacturing operations are also termed features. To distinguish between the two different types of features, “CAD-feature” may be used for CAD-geometry definition operations, while “manufacturing features” is the result of manufacturing

operations. It is necessary to distinguish between the two since this thesis deals with both CAD and manufacturing features.

### 3.2.1 CAD-formats and feature recognition

Unfortunately, all CAD-systems developers have created their own standards for creating CAD-features and constraints. This means that construction histories and parameters cannot be directly transferred between the different CAD-systems. A standard for the exchange of this information is under preparation, but it has not become industrially widespread yet (Kim, Pratt et al. 2008). Instead, a number of different so-called neutral formats are in use. These standardised neutral format standards allow the exchange of all types of geometrical elements: wireframe, surfaces, and solid bodies. However, the construction history and parameters are lost. Examples of neutral formats are: STEP, initial graphics exchange specification (IGES), and VDA-FS. A construction history created in one system cannot easily be reconstructed in another. This is true even though the geometry definition methods in different CAD systems show extensive similarities. Attempts have been made to create translators (Mun, Han et al. 2003) propose a macro-parametric language for the direct translation of construction histories. Translators are also commercially available from several companies, such as CADverter® (Theorem 2008) and FeatureWorks® (SolidWorks 2008). This software can reconstruct the feature history either interactively (by the user pointing out the features) or automatically. In addition to translating between construction histories, the neutral geometry can be scrutinized directly, identifying the features. This is known as feature recognition (FR). FR is the only available option when the original CAD-model lacks a construction history. The geometry is examined and parts of the geometry with presupposed purposes can be sorted out. However, the recognition process requires that features be reasonably “clean”. An example of this is when a cylindrical hole is identified by finding a cylindrical face with all surface normals pointing towards the centre. Had the normals been pointing outwards, the object would perhaps have been identified as a cylindrical rod. Complex feature interactions can generally not be resolved. However, research in the field is being conducted (Babic, Nesic et al. 2008). In FR, the neutral format CAD-model or alternative representations of it are searched using different techniques such as rule-based or by volume decomposition methods (Shah and Mäntylä 1995). Regardless of the method, FR requires criteria that can be used to separate the sought feature from the rest of the geometry. These criteria depend on the purpose and intent behind the feature.

The key motivation and the original use of FR is not interoperability between CAD-systems. Rather, it is finding features that can be machined so that the tooling paths can be automatically established see e.g. (McMahon and Browne 1998). This is referred to as computer aided process planning (CAPP).

## 3.3 AUTOMATING PROCESSES IN PRODUCT DEVELOPMENT

In PD, designers, production engineers and others perform intellectual processes that result in the geometry and other specifications of the product. Exactly what goes on in this process is difficult to grasp and involves a number of different types of activities. Apart from creative activities, such as invention, it also involves more repetitive tasks for which a straightforward procedure, possibly branched, can be established that does not require any creativity. With the different activities in the PD process on chart, it is possible to formalise them into a computer implementation. The scope of these computer implementations largely depends on the procedure to adapt the new design to meet the altered requirements. This also makes it possible to classify the systems. Such classification is given in (Claesson 2006). There, at the

lowest level called selection, it is just a matter of providing a number of variants of the products and selecting the one that comply the most with the requirements. When the requirements get more detailed, it is no longer possible to provide one variant for every conceivable set of requirements. Instead, the different subcomponents are selected and combined into a complete product. This is the basis of product configuration (Hvam, Mortensen et al. 2005). Configuration of products is a well established field, and commercial software shells for building configurators such as Tacton Configurator™ (Tacton 2008) are available. The configurator contains information on what combinations are valid. It also defines a process for configuring the product.

However, in some cases, the problem cannot be solved by only selecting a standard variant or combining variants of sub-components. In this case, a new design needs to be generated following an established route. It should involve not only general norms and standards for the design but also knowledge and experiences gathered from previous products. Trying to capture this process and its associated knowledge is the foundation of design automation (DA). (Cederfeldt and Elgh 2005) provide a definition of DA:

“Engineering IT-support by implementation of information and knowledge in solutions, tools, or systems that are pre-planned for reuse and support the progress of the design process. The scope of the definition encompasses computerised automation of tasks that directly or indirectly are related to the design process in the range of individual components to complete products”

From a DA point of view, the route followed when automating a design task can be subdivided into parametric and generative design. In parametric design it is possible to vary the parameters of a template CAD-model to comply with the altered requirements. In the case that the new design is outside the range of varying the parameters, a new design has to be automatically generated. This is then termed generative design.

The IT support mentioned in Cederfeldt and Elgh’s definition includes a number of computing methods such as KBS, CBR and others. These are combined in computer systems to automatically or in user interaction carry out the design process fully or in part. In order to structure the process prior to its computer implementation, methods from design science for structuring the design process such as dependency structure matrix (DSM) and function means tree may be used (Hubka, Andreassen et al. 1988; Ulrich and Eppinger 2008)

Further, the design process is seldom something static. It is constantly being developed in actively working organisations elaborating the product and process knowledge. The traditional way of capturing this knowledge for formalisation into a computer implementation is to let a “knowledge engineer” interview the domain experts. The knowledge engineer then gathers and structures the knowledge in a way that can be codified into the DA-system. This will produce a DA-system that applies the knowledge that was most recent and up to date at the time of the interviews. In order to keep the knowledge up to date, the domain experts would have to be interviewed again and the code updated. This is referred to as the commodity view on knowledge, since it regards knowledge as a commodity that can be captured and computer implemented. To reflect the dynamic nature of the design process, it is necessary to allow more active participation from the people involved in the PD-process. This is known as the community view (McElroy 2003). The community view can be supported by allowing the participants to extend and modify the knowledge contained in the program without requiring advanced programming skills and insight into the details of the code. Here AI tools, possibly in combination with the CAD-system can act as important help in gathering, managing and reapplying the knowledge with a degree of automation.

### 3.4 AI/KBS/KBE

The software commonly used in DA-systems can be divided into KBS and computational intelligence (CI) (Hopgood 2001). In KBS the knowledge is given explicitly and is ready to be applied directly. In CI, new knowledge is derived, for example, by employing optimization algorithms to drive the values of the design parameters towards given goals, given the design constraints. This is used when there is no simple analytic relationship between the requirements, the constraints and the design parameters. Instead, the goal is given and numerical experiments are used to find a solution that best fulfils the requirements on the design. Knowledge can also be obtained automatically by extending the knowledge-base as the work progress. For example, new completed cases can be added to a repository. This does not require any formalisation of the knowledge gained. The former cases that are most relevant for the new case can be retrieved by case based reasoning (CBR) (Maher and Pu 1997). The gathering of new knowledge is often termed knowledge acquisition.

In this thesis only the KBS representation of knowledge will be discussed. In KBS, rule-based systems are central. The knowledge is then given as explicit rules and facts. The rules are applicable if certain conditions expressed in the rule are fulfilled, such as when the value of a parameter is within a certain range. Fulfilling the requirement will cause the rule to trigger a reaction specified in the rule. When a rule is found to be applicable and starts a reaction, it is said to be “fired”. If the condition of the rule is examined but it is found that the condition is not fulfilled, the rule fails. In the example below, a production rule is given:

IF  $a < 10$  THEN size=small ELSE size=big

Should the parameter  $a$  have a value less than 10, the parameter “size” is set to small. Otherwise, it remains set to big. To represent engineering knowledge in real rule-based systems, the conditions and reactions are often more complex than in the example involving different types of computer support. Fulfilling the condition can e.g. trigger an FEA calculation from which the results provide values for other rules.

The rules are assembled in a rule-base that can be managed by adding revising and deleting the contained rules. The execution of the rules is in a rule-based system controlled by a so called inference engine which determines the order in which the rules are applied. There are different strategies in how to resolve the rule-base, of which the most important are forward and backward chaining. Forward-chaining starts from initial values and the rules are applied in the order in which they become applicable. Thus forward chaining is said to be data driven. In backward-chaining, the strategy is instead to arrive at predefined goals. This means that the starting point is the end values, and the inferencing engine determines an order in which to execute the rules so that the end values are fulfilled. Further, overlapping rules have to be handled. Otherwise this easily could result in an endless loop where parameters are first set by one rule and then reset by another so that the condition of the first rule becomes applicable again, causing it to fire and so on. This can be handled using meta-rules that e.g. say that the rule can only fire once when solving the rule-base. Consequently, the result will be governed by the second rule to fire. Another way is to assign priorities to the rules giving precedence to the one with the highest priority. An interesting example of prioritising in rule-bases is given in (Johansson 2008), where the priority is decided by the accuracy of the rule. The rules clearly overlap each other, but they are based on different methods with varying precision such as rules of thumb, analytic formula, and numerical simulation. It is the required precision of the end results that govern the prioritising among the rules in the rule-base.

### 3.4.1 Integration with CAD

The geometry of the product is as mentioned an important part of the product specification. If a rule-base is going to be used for automating the process of creating the geometry, it is not in all cases practical to first derive a set of design parameters and then make the CAD-model accordingly. The CAD-systems contain powerful functionality for geometry manipulation and evaluation. That functionality would be desirable to access as needed to obtain intermediate results from the CAD-system for use as input to the rule-base. As such, a close integration between the rule-based system and the CAD-system is clearly desirable. CAD-system developers have started to integrate AI functionality in the CAD-programs both representing KBS and CI. Examples of such systems are CATIA knowledgware® (Dassault-Systemes 2008) and NX6 Knowledge fusion® (Siemens-PLM 2008). The CAD-systems developers then claim to have integrated KBE functionality in their systems. The systems also provide means of representing engineering knowledge by providing expert systems shells so that rules can be defined to directly affect the parametric model. The engineering knowledge can also be represented as scripts, templates, MS Excel tables and so on. These can then be integrated into an automated process by invoking them using programs defined in application programmable interface (API) of the CAD-system. This provides the possibility of building design automation systems that allow the designer to work in a familiar environment. Further, the different types of knowledge represented can be accessed and managed by the engineers. The CAD-system is also, in many cases, closely integrated with the product document management system (PDM). The design team can, in collaboration with teams from production and other functions in the company, collaborate on knowledge management (KM) involving the build-up and maintenance of the knowledge content. The tools for KM provided in the CAD-system might not be sufficient for all functions needed. Integration is needed between CAD-systems and other applications allowing external programs to be created and integrated with the CAD-system.

## 3.5 APPLICATION PROGRAMMING IN CATIA V5

The experimental systems programmed to conduct this work have been made in CATIA V5. It is a hybrid modelling CAD-system, meaning that it has extensive functionality both for solid and surface modelling. In addition to the geometry definition functionality, it also integrates a large number of design analysis and synthesis tools. This allows direct manipulation and the evaluation of the model and its parameters. These functions can also be accessed and deployed using VB coding in minor programs, called scripts. The scripts are run hosted by the CAD-system. This allows the user to automate known parts of the work, rather than working interactively. Some examples are given in Figure 3.2 to provide an understanding of how the programming is carried out. All geometrical elements in the parametric and neutral format CAD model can be accessed, and routines can be defined to manipulate the elements. The programming Example 1 shows how to access all lines in the CAD models sketches and then scale them to their double lengths.

**Example 1**

```

Dim sketch1 as sketch
Dim sketches1 as sketches
Sketch1=model.sketches.Item(n)
For i=1 to sketch1.GeometricElement.count
  if TypeName(sketch1.GeometricElement.Item(i))=Line2D then
    sketch1.GeometricElement.Item(i).length=
      2* sketch1.GeometricElement.Item(i).length
  end if
Next

```

**Example 2**

```

Dim point1 as point
Dim Allvertices as vertex()
Allvertices=Model.search "Topology.vertex"
point1=AddPointOnVertex(Allvertices.Item(n))
Point1.GetCoordinates xyz
xyz(0) 'the global x-coordinate
xyz(1) 'the global y-coordinate

```

Figure 3.2 Programming examples in VB scripting.

The sketch in the model with item number n is first assigned to the variable sketch1 which is declared as the data type sketch. The geometrical elements in the sketch are thereafter accessed. These elements can be of different types (TypeName) such as spline and arc. If the element is of type line2D, its length is multiplied by 2. In the second example, Example 2 all the vertices in the solid-model are assigned to an array. A point is then attached to the vertex with item number n, and the global coordinates of the added point is thereafter read. Detailed coding examples are given in the appendix.

Macro programming and accessing and controlling CATIA from external programs has been essential for the creation of the experimental systems. Without the open API, the geometry handling capacities could not have been automatically deployed. This would have required developing a modeller with similar capacities, which would not have been possible. More documentation of the API is given in the CATIA manual on CD available from (Dassault-Systemes 2008).

### 3.6 FEA AND GEOMETRICAL IDEALISATION

The finite element method has gained widespread use in engineering analysis. It is used for solving a multitude of different engineering problems. Examples include linear/non linear stress and displacement, heat transfer, diffusion, and so on. The finite element method can be thought of as a numerical method of solving partial differential equations. See e.g. (Saabye Ottosen and Petersson 1992; Hughes 2000). The method has proved very applicable in engineering problems. This is because the designed products often have complicated geometry. The differential equations describing the studied phenomenon cannot be solved directly for the whole body of the product. Instead, the body is discretised into finite elements. The shape of these finite elements is given by node points for which the displacement is given by the formulation of shape functions. The differential equations can be approximated for the elements which allow e.g. the element stiffness matrix to be established. The element stiffness matrices are assembled into a stiffness matrix for the whole structure. The stiffness matrix provides a dependency between the displacements of the nodes and the force applied to the structure. The shape functions are formulated differently depending on the purpose of the element. Elements such as solid, beam, truss, and shell are all formulated to best capture the studied problem. In this way, the analysis is kept efficient, minimizing the computational work required to solve the problem. For example, if the studied region is long and slender and the purpose of the analysis is to study bending, the best selection is a beam element. The work presented in this thesis deals with thin walled structures that are loaded in both in the material plane and normal to it. Furthermore, the displacement along the shortest dimension, the material thickness, is negligible. These considerations make the shell element

the first hand choice. Die-cast and injection moulded parts may also contain regions that deviate much from being thin-walled. When that happens is that the displacements are of the same magnitude in all three dimensions. Thus, an element that takes this under consideration is required (i.e. some type of solid element). This will form a mixed dimensional FE-model where the connections between the meshes demand attention. This is because the elements have different shape functions and are not directly compatible.

### 3.6.1 Auto-meshing and mapped meshing

The process of discretising the structure into finite elements is commonly referred to as meshing. There are several methods for defining a mesh. It can be built up from start by extruding, sweeping, or revolving operations on lower dimension meshes that have to be built up from geometrical elements of a lower dimension than the resulting mesh. These meshes are generally referred to as mapped meshes. Mapped meshes give the user control over the created mesh, and the quality of the mesh is generally high. The downside is that, it is a time-consuming way to define the geometrically complicated meshes that are often needed in engineering problems. Another emerging method already included in most commercial FEA software is auto-meshing. In auto-meshing previously defined geometry is used as a target for the mesh. There is a multitude of different techniques for auto-meshing. One well-known method for tetrahedral and triangular meshes is the bubble mesh method. The method is based on placing spheres at geometrical entities of the model. The spheres are distributed evenly over geometrical entities of the model. The element nodes of the finite elements are placed at the centres of the spheres. Recently, (Su, Lee et al. 2004) proposed a methodology to automate the meshing of mixed dimensional FE-models. A mix of solid and shell elements is often required for the Die-cast geometries studied in thesis. The automated mixed dimensional meshing is realized by modifying the original meshes to comply with the constraints between them. According to (Su, Lee et al. 2004) the original meshes can be constructed using number of techniques. Those techniques can be classified under three main categories: block decomposition, superpositioning, and the advancing front methods.

The resulting auto-mesh is generally of lower quality than the mapped mesh. On the other hand, it can be defined faster. The input for the automatic mesh generators is geometrical models that have to be idealised prior to meshing in order to make a purposeful FE-model. The idealisations are of two types (Dabke, Prabhakar et al. 1994). The first is the global idealisation. Its purpose is the removal of all details not relevant for the analysis so that these do not disturb the application of the mesh and the results. The element idealisation is the dimensional reduction of the solid geometry into lower dimensional elements, thus making a more purposeful model. This is the motivation for idealising the geometry of CAD-models prior to auto-meshing. Long and slender regions may be idealised into wire-frame, while at the same time removing small details such as screw attachment points. A thin walled structure may be idealised into its mid-surface in the same way. If the purpose of the FE-model is to capture the overall mass/stiffness distribution of the structures, small fillets, holes and so forth may be removed. This is because they contribute very little to the mass/stiffness of the structure. There is a variety of methods for automating the geometrical idealizations prior to the auto-meshing. They are based the direct algorithmic processing of the geometry in a way that allows for implementation in general purpose FEA packages or by the use of different types of expert systems. These two major strategies and how they relate to the work presented here will be elaborated in the next chapter, related work.

### 3.7 DIE DEPENDANT PRODUCTION PROCESSES

This thesis involves FR of manufacturing features. In order to understand how it works, it is necessary to provide the reader with an insight in the details of these two production methods. The geometrical restrictions are most accentuated in the PM process. However, the die-casting process also shows some restrictions that are used in the experimental systems. A general description of the production processes with special focus on the geometrical restrictions is therefore provided in this chapter.

#### 3.7.1 The die-casting process

In die-casting a tooling cavity is injected with molten metal under pressure. Mostly light alloys such as aluminium, zinc, and magnesium are die-cast. The die is permanent so it needs to be parted into two tooling halves in order to eject the ready part. These are called cover and ejector dies. They are held in place by one stationary and one moving platen attached to the die-casting machine. The machine opens and closes the die by moving the moveable platen. The dies are replaced when another product is to be manufactured. Sufficiently high pressure must be exerted to keep the tooling halves pressed together during the injection of the metal. The zone where the two halves meet is referred to as the parting line. Drafts are required especially on surfaces facing each other so that the part shrinks towards them to enable ejection of the part. The drafts needed are typically between 1-5°. The ejection of the part is accomplished by pins controlled by the casting machine. Further, the cycle-times are kept low since high productivity is needed to carry the high investment. For this reason, the walls of the part are kept thin and reasonably equal in thickness. To fully use the strength of the material, parts are typically designed with a structure of reinforcement ribs. These ribs are formed by grooves in the tooling dies. When placing the reinforcement ribs, the part should be designed so that the shrinking effect does not introduce any warping and skewing of the part. In addition to the geometry oriented in the tooling draft direction a transverse “floor” can be formed if a gap between the tooling dies is leaved when the tooling is in closed position. The orientation of this parting line feature can be allowed to vary as long as it does not block the tooling opening direction. These restrictions lead to that the part can be regarded as a structure of features formed in the tooling halves running in the tooling opening direction, possibly disrupted by material formed in the parting line gap. The picture below shows an imaginary die-cast part displaying both types of features.

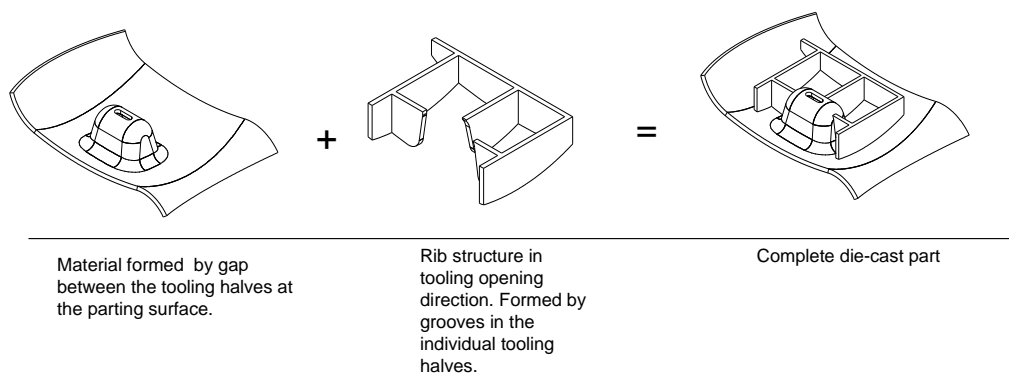


Figure 3.3 Tooling direction and parting line features. (Illustration from Paper VI)

In addition to these types of features, the tooling assembly can also be equipped with side actions that can form features such as holes in other directions. For cost and production



reliability reasons, the number of side actions should be kept as low as possible. For a more elaborate discussion on the die casting process, see, for example, (Andresen 2005).

### 3.7.2 The features of the die casting process

Several different types of features will result from the process. These will be used in this thesis to enable feature recognition in the die-cast parts. The parts of interest may be described as ribbed shell structures, where the rib feature and the shell feature are the most important ones. The shell feature is created in the gap between the dies and can have an arbitrary shape. Complex double curved shapes are possible as long as they permit the tool to open.

The rib feature can start on any surface. However, once it has started it must continue up to the shell feature. Rib features can be present on both sides of the shell feature. Alternatively, if no shell feature is present, the rib feature will continue through the whole part. Discontinuing the rib feature would mean that there is an undercut present. The undercut must then be formed by a side action in the tooling. In addition to the rib and shell features, there are also sometimes areas present that cannot be directly identified as rib features since they do not have to have one dominating dimension in the plane (see Figure 3.4). Since the section is arbitrary, it is not possible to standardise it. Therefore, the user has to define it depending on the type of product being designed. Still it has to follow the restrictions of the process. It extends in the tooling draft direction and is formed in the die. The section area must be limited. Otherwise, a big accumulation of material would result and this would not comply with the low cycle time requirement. Here, this type of feature is called a user defined feature. The picture below shows a die-cast part displaying the typical die-cast features.

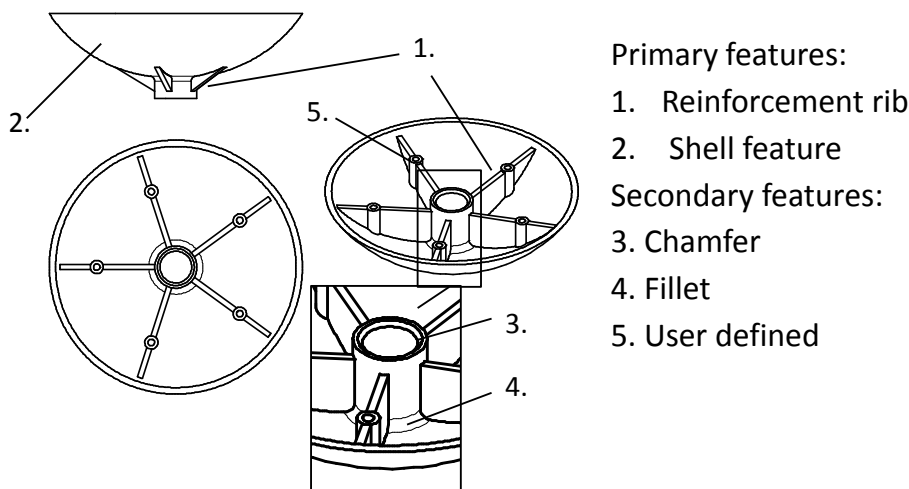


Figure 3.4 Features of the die-casting process (Illustration from paper VI).

### 3.7.3 The PM pressing and sintering process

Powder metallurgy is a vast field involving many different types of materials and production processes. The base material is metal powders. They are manufactured by different processes such as the iron sponge method, milling or by gas or water atomisation. The powders are used in many different production processes, such as die pressing, isostatic pressing, powder injection moulding, powder metal forging, and spray forming (Ashby 2005) each having its own types of products.

This thesis is only concerned with the die pressing and sintering processes. Examples of components produced by the process are those where a porous structure is needed, such as self lubricating bearings and filters. The process is also currently advancing into producing parts subjected to substantial stress and wear such as synchronizing rings and gears competing with processes like forging. It is this type of components this thesis focuses on.

The main attractions of the process are the formation of net-shaped parts with good dimensional stability. The production equipment represents a substantial investment, which requires a high volume of parts. Good economy is obtained in that few secondary operations are necessary. The process also provides good material utilisation.

In this process, the metal powder is filled into a die and compressed to high pressure, typically 600MPa for ferrous powders. This causes plasticity in the powder grains making them adhesively bonded together to such an extent that the part can be ejected from the die without breaking. The part is often then referred as being in a “green” state. In order to make the part homogeneous and acquire sufficient mechanical properties, the green part is heated up in a furnace with a controlled atmosphere to a sufficiently high temperature to allow the grains to merge, forming “bridges” between them called sinter necks. These necks grow until the material is almost homogenous, while leaving a small residual porosity in the part. The porosity is primarily due to the necessity to add a lubricant to the powder in order to compress the powder sufficiently. The lubricant occupies a certain volume in the part and is burnt off prior to sintering, leaving a residual porosity. Porosity and inhomogeneous distribution of the porosity is also due to the fact that there is still high friction internally in the powder and against the die walls despite the presence of the lubricant. The further from the punches, the lower the pressure becomes. This can be counteracted by increasing the pressure and adding more lubricant. However, there are limitations on how high the pressure can become, considering both the tooling material and the powder itself. Compacting pressure that is too high is likely to cause either cracks in the green part when ejected or impaired tooling life. Increased lubricant volume will also mean more porosity since it occupies a greater volume.

### 3.7.4 Tooling technology and design considerations

The pressing equipment consists of a number of tooling elements. These must move independently of each other to obtain as high and evenly distributed density as possible. The tooling motions are realised by fitting the tooling elements on platens which are, in modern production, individually controlled by a hydraulic press. Ideally, the parts should be designed with a number of levels, each formed by a separate punch to maintain high and equal density in the whole part. This is not always economically or technically possible. The number of punches must be kept low (typically three lower and two upper) in order not to make the tooling assembly overly complicated and thereby expensive and unreliable in production. According to recommendations found in handbooks such as in (Mosca 1984; MPIF 1998; Höganäs 2007), minor features can be permitted if their height is less than 20% of the total powder column height. Here, a feature formed by an indented or protruding punch face is called a face-form. Due to the friction in the powder, the diameter/height ratio of each powder column must be kept within certain limits. Yet, the height/diameter ratio cannot be too small or the part would break when ejected. Further, there must not be any protruding sharp edges in the parts. Otherwise, the tooling members to form them will be excessively fragile.

Corners should be rounded to increase the powder flow while filling. There should not be any small radial distances, which mean that tooling elements with small section areas have to be

used. These tooling elements could easily deform and jeopardize production robustness and cause dimensional instability of the parts.

### 3.7.5 The features of the PM process

As in the case of the die-casting process, the PM process has typical features as a result of the shape restrictions of the manufacturing process. A section, with planar ends, that is pressed by a tooling element can be called a powder column. The PM part can be seen as mainly consisting of such columns stacked on top of each other or placed side by side. The process also allows secondary features to be formed on the powder columns. The faces of the punches may be indented or protruding, provided that the height of them is less than 20% of the total column height (as mentioned in the former section). Further, chamfer and radii can be placed on the powder columns, provided that they are not placed on an outer edge so that the punch must be manufactured with a fragile edge. To conclude, the PM part can be decomposed into a limited number of manufacturing features as shown in the Figure 3.5 below.

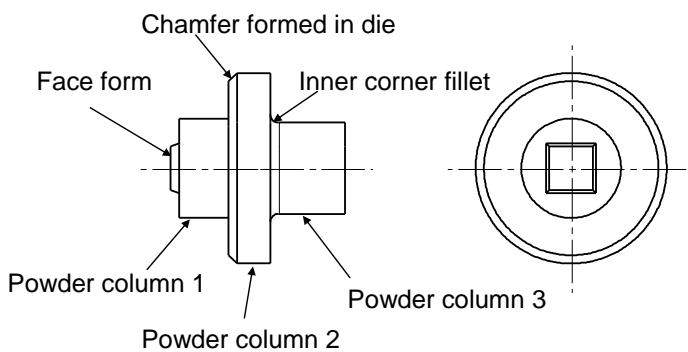


Figure 3.5. The features of the PM process. (Illustration from Paper V)

These shape restrictions are further described in Paper V and will be utilised for the automated FR needed in the experimental design advisory systems.

## 3.8 CATEGORISATION OF THE MANUFACTURING FEATURES

To summarise the above, the features that can be formed by the two production processes fall into three main categories: primary, secondary and specialised. The primary features are those that define the basic shape of the part and the secondary features modify it. The specialised features can have any shape, but they still have to comply with the process restrictions. The primary features of the die-casting process are two, the shell and the rib features. The PM process has only one primary feature, the powder column. The basic shapes of the two processes are typically complemented with secondary features such as drafts, chamfers, fillets and rounds. There are also limitations in the PM process on where to place the fillet, round and chamfer features. As mentioned, they must not be placed in outer corners, which would result in tooling elements with sharp edges.

Both production processes also allow specialised features, here called user-features for the die-casting process and face-forms for the PM process. These features can be of arbitrary shape as long as they comply with the process shape restrictions. For the die-casting process, the user-feature can have any shape of its section as long as the section area is limited and the feature is oriented in the tooling draft direction to allow ejection from the die. For the PM process, the face-form can have an arbitrary shape. However, its height must be limited compared to the

height of the powder column and it must also allow ejection from the tooling. Table 3.1 on the next page show the different features.

Process \ Feature	Primary feature	Secondary feature	Specialised features
PM Pressing and sintering	Column feature	Chamfer, round, Draft	Face form
Die-casting	Rib feature, shell feature	Chamfer, round Draft	User defined Example: Screw attachment

Table 3.1. The features manufactured by the two processes.



## CHAPTER 4

# RELATED WORK

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## CHAPTER OUTLINE

This chapter reviews the related work both in terms of design advisory systems for PM parts and for the automated idealisation of the parts prior to FEA meshing.

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The different technologies described in the previous chapter are used in computer systems to support the development of products by reapplying general and specific knowledge from previously designed products. Such systems are widely used in numerous applications. Since the application examples presented in this thesis evolve around systems for the automated geometrical idealisation of CAD-models and design advisors for PM parts, an attempt is made in this chapter to cover the research frontier and industrial practice concerning such systems. It should be noted that automated geometrical idealisation is a much wider area than design advisors for PM parts. This is not at all surprising since geometrical idealisation is much more generally needed and applied whenever CAD-models need to be prepared for FEA. The PM industry meanwhile, is a specialised and process centred branch. The chapter starts with work related to geometric idealisation and proceeds to design advisory systems for PM parts.

### 4.1 AUTOMATED GEOMETRICAL IDEALISATION

The purpose of the idealisations is to prepare the geometries for FEA. There are basically two major working principles: The algorithmic approach and the KBE approach (Paper I). In the former case, general idealisation tasks are identified and complete software package are provided to meet this demand. One example of functionality is the partition of the geometry into different regions, each with geometry most suited for meshing with a certain type of elements (e.g. separating thick geometry for meshing with solid elements from thin geometry for meshing with shell elements). Another commonly used functionality is conditioning the geometry prior to meshing. The CAD-data may contain small details irrelevant to the analysis. However, if the geometry is to be meshed directly by auto-meshing the mesher would try to resolve these areas, resulting in a less purposeful mesh. Consequently, the software has to aid in the identification and removal of these features. This is known as de-featuring.

The latter approach, the KBE, is much more versatile and process oriented. It employs the use of ES, KBE and other AI techniques to make the system flexible so it can be adapted to the specific task and be maintained as the knowledge increases. The resulting systems must directly influence the geometrical model. For that reason, the representation of the model becomes important. This is shown in several publications utilising the features of the CAD model, presented in the next section.

KBE systems tend to have and have a broad focus (being more of a DA system) rather than having the narrow scope of merely preparing the geometry for meshing. Notably, the KBE systems can adopt algorithmic procedures to perform tasks needed in the process, and the

algorithmic approach may contain KBE elements. Related work from the two different approaches is presented below.

#### 4.1.1 The algorithmic approach

The algorithmic approach is well established and high end FE –packages often contain functionality for this purpose. Examples of such software are HyperMesh™ (Altair 2008) and AI\*Environment™ (Ansys 2008). In Paper III, a comparison is made between the two software packages regarding their performance when automatically extracting a mid-surface from a number of solid geometries.

The theoretical background for this software is seldom explicitly given in the programs' documentation. It is clear however that the Medial Axis Transform (MAT) is used in geometry processing. In 2D, the MAT may be described as the path of the centre of a disc of the largest possible diameter inscribed in the geometry at a given location as it moves to all possible locations in the geometry. The MAT also has a three dimensional counterpart, where the disc is replaced by a sphere moving to all locations in a volume. The research on the use of MAT for idealisation purposes has been conducted in e.g. Queens University of Belfast (QUB). Its main object is to identify regions in CAD-models that are of idealisation interest (Armstrong, Donaghy et al. 1996). The ratio between the medial axis length and the radius of the disc is checked at specific locations in the model so that small details irrelevant to the analysis can be detected. These can consequently be removed from the model. The actual removing is difficult to accomplish automatically since it requires the reconstruction of the model without the feature. Similarly, regions for meshing with certain types of elements, such as long slender regions for meshing with shell elements, can be detected using the MAT (Armstrong, Bridgett et al. 1998). Errors in models that can cause problems when trying to automesh them can also be detected using the MAT. This is elaborated in (Lee, Price et al. 2003).

Idealisation may also mean reducing thin 3D solid CAD-models into shell element models. In doing so, it is possible that the creation of the mid-surface is required. A special surface pairing algorithms proposed by (Rezayat 1996) is commonly used. Basically, the algorithm searches the CAD-model and finds surfaces located opposite each other by searching in the normal direction of the surface. Whether the paired surface really represents a thin-walled section is checked as follows:

$$\min(\mathbf{L}, \mathbf{H}) / t > X$$

where L and H are the length and height of the surface and t is the distance to the opposite surface. Specified by the user, X serves as the criteria for the thin-walled section. Next, the edge faces (i.e. the common faces between the paired faces) are identified. The paired surfaces, the edge faces, and some additional geometrical information are used to build adjacent graphs. Those graphs serve to structure which surfaces are paired and which edges serve as boundaries. The actual mid-surfaces are constructed so that they interpolate a set of points that have been created mid-way between the surfaces of the pair. The grid density of the point set is adapted to the curvature of the surfaces. The created mid-surfaces are consequently an approximation. Further, they are independent of the original surfaces' parameterisation. The created mid-surfaces are sewn together using the adjacency graphs and operators to extend and trim the neighbouring surfaces where possible. Alternatively, new surfaces are formed between the patches. In this manner, a trimmed mid-surface representation of the whole CAD-model is formed.

It is in some cases possible to treat the extraction of the mid-surfaces as a matter of parameterisation of the CAD-model such as is shown in (Fischer, Smolin et al. 1999). Here, the mid-surfaces are extracted from simple geometries by re-parameterising the NURBS curves of the geometry.

#### 4.1.2 Expert systems used for geometrical idealisations

Early attempts at using KBS to automate FEA have been complemented by other CI techniques (Manevitz and Givoli 2003). Examples include genetic algorithms and neural networks. This is motivated by the fact that some of the knowledge required for automating FEA is not suited to be represented by rules. This has been acknowledged by (Li and Qiao 2003). They show how a rule-based ES can be used for the automated analysis of aircraft frames. The ES rules can successfully be used for tasks like selecting the correct element type for a given geometry. However, they fall short in classifying the structure type from digitised drawings. To tackle this drawback, the KBS has been complemented with a neural network that has been trained to classify the parts correctly, enabling the automated analysis of all parts.

#### 4.1.3 Feature supported geometrical idealisations

The features of the CAD-model can be used to support finite element idealisations, as shown by (Dabke, Prabhakar et al. 1994). A system is presented that bases the idealisation on the features in the CSG. The system provides both global and element idealisation by using two different ES. First, an ES called DESIDE-X (Design Idealization Expert) analyses the features of the CSG tree by applying a rule-base that can conclude whether the feature is relevant. This is done by calculating the mass of the feature and establishing a load path through it while at the same time considering the functional intent behind the feature. This information, and also the engineering significance of the feature, is used as input to the backward/forward chaining expert system to decide whether to keep or remove the feature. If the feature is removed, the CSG tree is updated. The rules in the system have been acquired by interviewing analysis experts. The rules contain general knowledge on FEA-modelling from a number of different applications. One such is the extraction of natural frequencies in structures. After removing all non-relevant features, the next step is to apply a tool called ADVANTAGE. ADVANTAGE has an ES as well. The ES suggests appropriate element types by classifying the features by their shape and the way they are loaded. The geometry is converted and sent to the analysis package for evaluation.

(Belaziz, Bouras et al. 1999) also present a feature-based approach. However, the approach uses low-level features that do not have any direct engineering significance such as faces, edges, and vertices. The CAD-model is morphologically analyzed and represented in a way that allows the idealisation of the geometry. An important contribution is that the geometry can be accurately reconstructed from the representation.

#### 4.1.4 KBE approach

As mentioned the KBS approach is process-oriented, having a broader scope of automating the PD process. One way in which KBE is used is by adapting high level elements to the design specification. The resulting design can automatically be analysed from various perspectives by the system. Examples are found in (Chapman and Pinfold 2001) and (La Rocca, Krakkers et al. 2003) who describe high level design systems for specific products. The engineering knowledge, including the procedure to idealise and analyse the designed structure, is reapplied in the new PD project. This, in a sense, provides automated FEA within the permissible variation between the current and former products. (Chapman and



Pinfold 2001) show how to automate the analysis of a passenger car structural body using the KBE system ICAD. The design obtained is idealised by reusing the procedures of analysis that have been established in former projects. Special care is taken concerning the automated idealization of the structural members. There, rule sets can be applied to idealise them into surfaces or wire-frame target geometry for subsequent automated beam or shell meshing. Properties such as the bending resistance of the structural members are automatically calculated and fed to the mesher. The joints between the structural elements are automatically classified as spot-welded or bonded for example. A rule set determines how the joints should be represented in the FE-model, depending on the surrounding material. A similar system is described in (La Rocca, Krakters et al. 2003). They present an automated design system that allows the automated structural analysis of an aircraft. Also (Isaksson 2003) presents a system that has been employed in the aircraft industry called knowledge-based integrated design and evaluation system (KIDS). The system consists of a nearly ready conceptual product model of an aircraft engine, including a well-established analysis model of it. The conceptual model can be revised to meet the new product specification proposed by the customer in a fraction of the time spent in a normal PD project. This allows the creation and evaluation of a high number of design suggestions. (Boart, Andersson et al. 2006) demonstrate the use of knowledge enabled pre-processing for making an element idealisation, mesh generation, and the completion of an input record for solving in an FEA solver. It is demonstrated on an aeronautical component for transferring the propulsive forces to jet engine to the wing of the aircraft. The motivation is to enable the analysis of an increased number of component configurations. The time saved using the KBE system (as opposed to interactive work) increases with the number of component configurations studied. The outcome will be a more thoroughly studied design in the same time frame. The KBE approaches demonstrated by Isakson and Boart, Andersson et al. are not based on having a template model from the start and merely varying the values of these parameters. The combination of the parameter values can easily lead to an update error in the model. Instead an instance of the design to be studied is created by the KBE system according to the specification and analysed thereafter.

## 4.2 SYSTEMS TO SUPPORT THE DESIGN OF PM PARTS

The second application example involves systems for supporting the design of PM parts. There are to the author's knowledge no such systems of any considerable importance commercially available. Minor computer implementations exist such as MPIF "electronic design guide" available from (MPIF 2008). In addition, educational resources such as Leonardo (EPMA 2008) and e-learning from MPIF are available. However, they are intended to provide basic education and do not involve and automated design or design evaluation capabilities of the PM part. In research, however, reports on automated PM design support systems are found. Most of these are based on specialized CAD-systems that are used to guide the designer through the process of defining the geometry, ensuring that the resulting geometry successfully can be manufactured by the process.

### 4.2.1 Automated support for design of PM-parts

A system with the purposes of guiding the designers is reported by (Smith 2003). The system, called interactive design advisor (IDA), was created by LISP programming in AutoCAD. It presents a number of typical PM features to the designer such as cylinder, gear, and key hole. The user can add these features to the CAD-model in a design-by-features approach. The features are parametric, and the dimensional parameters are simultaneously set by the designer. The parameters settings and the combinations of features are subsequently evaluated by a rule-base operating in conjunction with the CAD-system. This provides a

manufacturability check after adding each feature, ensuring that the completed design can be manufactured. There have been similar attempts, such as is reported by (Dissinger 1995). Rather than providing complete features, a specialised sketch pad has been created allowing the user to sketch the sections of the powder columns in each level of the part. It utilises the fact that a PM part must be designed in a limited number of levels to be pressed (such as is described in the former chapter). The user sketches the section of the part in each level. The sketched section is then extruded to form a solid model. The suitability of the sketched section is evaluated as well as the solid section. The evaluation is made against a rule-base containing the rules and recommendations for making a PM part suitable for production. There are also rules with the purpose of securing the validity of the CAD-model. They check, for example, that the sketched profiles do not self intersect or intersect each other.

#### 4.2.2 Automated tooling design

To a large extent, the properties of the part are dependent on the pressing equipment and the design of the pressing tooling, see e.g. (Beiss 2007). Therefore, the design of the parts and the pressing tooling are closely related. The rules and recommendations used in Dissinger's and Smith's systems must be valid for all types of products, materials and pressing equipment. Consequently, a large safety margin must be added. As a result, the design of the parts should be done simultaneously as the design of the pressing equipment to obtain an increased precision in the system (Paper V). There are reports of automated tooling design systems such as (Zenger, Kim et al. 1995) that support the tooling design process. However, the starting point is that the design of the part is already accomplished and the purpose of the system is to aid the design of the pressing tooling. This involves planning the tooling configuration to compact the part. This is done interactively by the designer selecting geometrical elements in the part. With this determined the system aids in typical tooling design activities. Examples include choosing the correct tolerances for the tooling elements, dimensioning them against mechanical failure, and determining the correct lengths of them so they can be fitted in the adaptor considering the whole pressing cycle. These activities are supported by retrieving process data from a number of different databases containing e.g. suitable tolerances for the elements. The databases are updated to reflect the increased knowledge from the ongoing production of parts.

#### 4.2.3 Design by features and feature recognition

All the presented systems are intended for interactive work. If one wanted to run the system automatically when, for example, re-evaluating previous design to verify and possibly refine the knowledge contained in the systems, or as an integral of an automated optimization loop, then these types of systems cannot be used since they represent the design by features approach. Evaluating former designs would instead require a feature recognition approach. No reports of such systems applied to the PM process have been found. However, in the work of (Lockett 2005) a feature recognition approach involving the injection moulding process is made. The starting point for her system is neutral format CAD-models containing a design suggestion of parts intended for manufacture using the injection-moulding process. The systems start by identifying features in the parts by implementing a FR procedure involving the prior extraction of the mid-surface. A rule-based FR on the graph representation of the surface is made. It is easier to accomplish FR on the mid-surfaces rather than on the solid-model directly, she argues. After identifying the features, a rule-based evaluation of their manufacturability is made. The rules are based on design recommendations from suppliers of plastics and tooling technology.

### 4.3 THE SCIENTIFIC AND PRACTICAL CONTRIBUTIONS IN THE CITED WORK

The cited work is concerned with specific applications, some for PM others for injection moulding and so on. The research projects seem to have been directed towards the applications. In this thesis, two different applications are studied, PM and die-casting. This provides the opportunity of studying the commonalities between design supporting IT systems as will be accounted for in the next chapter.

The motivation behind the cited work is to reduce the time spent on modelling the problems for their subsequent analysis, reducing the time spent in the whole PD process which was described in Chapter 1. Ultimately, it will have to be assessed to what extent the proposed systems shorten the PD process. This has not yet been done. The reason why may possibly be that the systems are not mature enough. In order to evaluate their contribution they have to be used in actual design work so that comparisons between them and alternative methods can take place. This would also mean that the work, which up till now has been of an explorative nature, can get more detailed. Once a consensus has been reached in the scientific community on how to construct the systems, the character of the experimental systems will change. Following the DRM, the studies will be directed to addressing details on the construction of the systems rather than proposing complete systems. An example of a limited, but yet important building on previous results is presented in this thesis. The construction of PM parts have been adapted from the work of Dissinger and adapted to an FR approach.

### 4.4 REQUIREMENTS FOR THE SYSTEM STUDIED IN THE THESIS

In conducting the work and becoming gradually more acquainted with related research, several issues in addition to the research questions have emerged as being important to address. As of yet there does not seem to be any attempt to address all die dependent production processes in a common framework. Further, little information is given on how the systems can be kept up to date to reflect the knowledge build-up in the organization. The reviewed systems include databases in which the information can be revised. However, they do not elaborate on how they should be maintained to reflect the increased knowledge. As earlier stated, product and process knowledge increases in an active PD organization, and the system must be able to support this activity. Further, the development of computers and simulation technology has made it possible to optimise products to a larger extent. In order to take advantage of this, the systems must be able to operate without user interaction. The first papers (Papers I-II) presupposed that the models had to be created in a certain CAD-system using its native modelling commands. It was found that this was unreasonable since the advanced die-cast geometries could not be modelled using only a limited number of simple commands. Further, restricting the modelling process would negate the possibility of using the functionality in the CAD-systems for efficient modelling. Further, the usefulness of the system is multiplied if it is possible to operate on arbitrarily defined CAD-models. Consequently, this became an important issue to address in the following papers. Below is a summary of the issues that the studies should focus on in addition to the research questions in the present and future work

- Address DFM and other downstream requirements for all die-dependent production processes.
- Focus on the KM process and the integration with existing IT tools.

- Provide possibility to operate autonomously to enable automated optimisation loops and automated knowledge acquisition.
- Not be dependent on that the design originates from a certain type of CAD-software.

The above is to be seen as providing a direction for the work, highlighting issues that emerged as important while conducting the work. To address the research questions (and also considering the above), a parameter and feature structure specialised for the production process was developed. It is based on the manufacturing features previously described, in Chapter 3.8 that emerges from the production processes shape restrictions. The manufacturing features can be reconstructed in the target CAD-system using CAD-features corresponding to the manufacturing features. The structuring of the models into features provides a purposeful parameterisation of the elements so that they can form a starting point for evaluation tasks and DA. Having the CAD-model reconstructed in a target CAD-system will provide access to the synthesis and analysis tools provided by the CAD-system. This will allow the designer to work in a transparent, user manageable, and familiar environment which can be handled in the existing IT infrastructure in the company. In order to automatically derive the feature structure, a specialised FR technique based on identifying and parameterising the features of the neutral files is developed.



## CHAPTER 5

# CAD-MODEL PARSING

## CHAPTER OUTLINE

This chapter presents the results of the work. It explains how the IT-system should be constructed to support the design process at different levels of detail and user proficiency. It will describe the automated interpretation of the neutral file CAD-models, and how to reconstruct them with construction histories tailored for the subsequent use. It will show the intent and scope of the system. Actual applications will be presented in the next chapter (Chapter 6), whereas the generally applicable reconstruction method will be described in the current chapter.

The process of automatically deriving a parameterisation and construction history for the purpose of geometrical evaluation and design automation was introduced in Chapter 1. The process is realised by FR based on typical features for each production process. When the features have been identified, they are reconstructed considering the subsequent use of them. Parameterisation and construction history will facilitate the following activities. The process requires prior knowledge on both the product and its production process. This knowledge has different degrees of generality. See Figure 5.1 below.

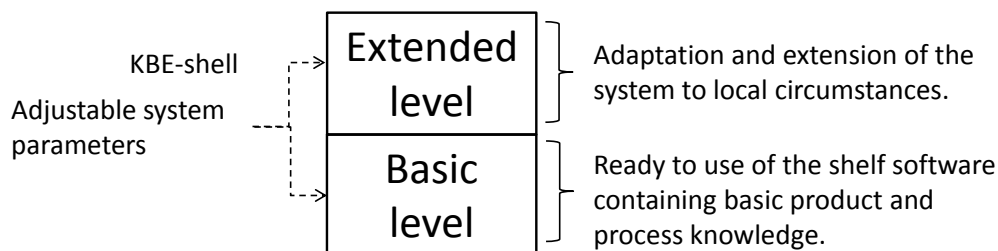


Figure 5.1 system levels.

At the basic level, there is the knowledge unlikely to change unless the working principal of the process is changed. This type of knowledge can be represented as long-lasting recommendations of how to design for the production process, and is applicable to all parts. There are at this level also similarities between different families of production methods. The die-dependent methods have the restriction that the geometry must be such that the die is able to open and eject the ready part (no undercuts in the geometry). There are probably similar restrictions for other families of production methods such as non-permanent mould methods sand and investment casting and between different machining operations. The basic knowledge by itself will only allow the systems to support the design process at the rudimentary level. Yet, it can serve as an important manufacturability check, ensuring that none of the basics are overlooked in the design, or possibly replace thin-walled entities with surface representations. On the basic level, it is possible to provide off-the-shelf systems with the basic knowledge encoded for general use. In order to increase the precision, scope, and detailing of the design evaluation and automated design, it is necessary to allow adaptation to the local circumstances. Geometrical recommendations vary depending on the type product

being developed, thus providing some of those circumstances. The material, tooling material and production equipment may be of another type than was anticipated in the recommendations. In order to adapt to this it is necessary to provide the possibility for the user to add and revise the knowledge. The threshold parameters given in the basic level can be adjusted to reflect the specialisation of product, material and production equipment. In addition to merely tuning the thresholds, it is necessary to provide possibilities to add special feature types (user-defined and face-forms as explained in section 3.8). These features must be identified, and which reactions should be triggered once they are reconstructed needs to be specified. This part of the system can never be provided as a ready-to-use, off-the-shelf system. Instead, the KBE shell provided in the CAD system can play an important role since it allows the tailoring of the system in a user-familiar environment. It is envisioned that a basic program package can be provided covering the basic level and that it can be explicitly read and adapted by the user as needed for more advanced use.

## 5.1 READING THE GEOMETRICAL ELEMENTS

The API of the CAD system permits, as earlier mentioned, access to low levels geometrical entities of a CAD-model, imported from an arbitrary modeller. Geometrical elements included are the boundary elements of the solid body (i.e. the vertices, edges and faces) and also wireframe and surfaces included in the import can be accessed as individual objects. Their properties, such as the coordinates of the object vertex are accessed by attaching points to them, such as is shown in the example given in Section 3.5. Likewise, the coordinates of the endpoints and intermediate points on the edges of the models can also be found in similar ways. Points are placed on the edges of the models. From their positions in the CAD-models, global coordinates, parameterised lines, arcs splines, conics and other elements provided in the modeller can be used to interpolate the points, thereby creating a parameterised wireframe entity approximating the edge. For the purpose of making the demonstrators in this thesis and for simplicity, only arc and line entities have been used. Consequently every edge in the imported solid-model has been fitted with three points, the endpoints and a point located at the mid-point of the edge while travelling along it between the endpoints. The Figure 5.1 below shows a model edge to which the three points have been attached. If the ratio between the distance between the mid-point and the line between the endpoints i.e. the height of the segment,  $h$  and the chord length,  $c$  ( $c/h$ ) is below a threshold value a parameterised line is created between the endpoints. In the other case, an arc through the three points is created. Note, that the wave shaped the edge in the bottom of the figure wrongly would be identified as a line. However, the types of geometries studied have not contained any edges of this type. For this reason, the simple arc-line resolution method has been found sufficient. If higher resolution is required, the number of points can be increased.

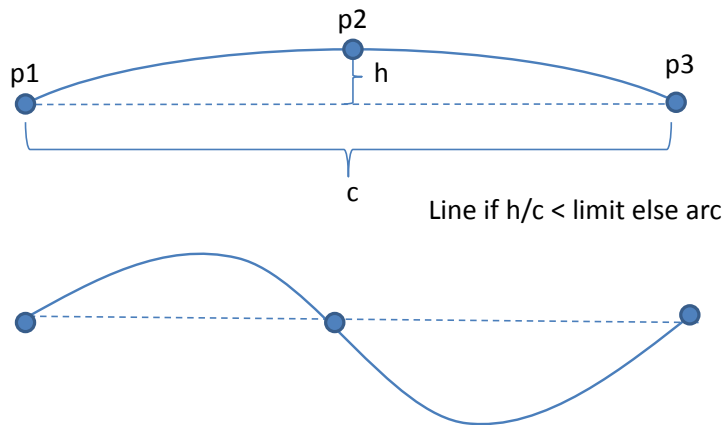


Figure 5.1 Fitting parameterised wireframe to model edges.

In the modeller, edges describing full circles are divided into two semi-circles so that a situation where the end-points coincide does not occur. The two semi-circles are in the resulting wireframe merged into a complete circle. In the way shown parameterised wireframe geometry can be fitted to all edges of the imported solid-body. In practice, only the wireframe geometry needed for the purpose of the reconstructed solid model is created in the target modeller. Also other geometry than just the edges may be needed such as boundary elements in the solid (e.g. planar faces). They can also be retrieved in the model and consequently used as references for adding parameterised elements, such as when creating the planes needed as references for the sketches.

## 5.2 CONSTRUCTING GEOMETRY FROM THE PARAMETERISED LINES AND ARCS

Having parameterised wireframe geometry provides the possibility of using it to construct associated geometry. A line segment can, for example, be used to create an extruded surface, and a prismatic body can be extruded from a closed loop. Depending on the subsequent use of the model automated routines can be employed to reconstruct the original solid geometry or possibly construct alternative representations of it. The reconstructed model has a parameterisation and feature structure that is ready for applying automated routines to it for DA and geometry evaluation purposes. Such parametric models are therefore in these thesis called DAR (Design Automation Ready) models. The DAR-model may be impractical to create interactively. However, since it is automatically derived from any CAD-model sharing the same production process, it can be quickly obtained and can be integrated in the product development process. The use of the DAR-model for specific purposes will be explored in the following chapter, application examples.

## 5.3 USING THE DAR-MODELS

In the application examples that will follow in Chapter 6, die-cast and PM pressed and sintered parts have been elaborated. Just the same, it is likely that the same method can be used wherever there are geometrically constrained geometries. It is also likely that successfully defining such systems is easier the more constrained the process is.

The procedure of creating the DAR-model and their subsequent use in PM and die-casting is depicted in Figure 5.2 below.



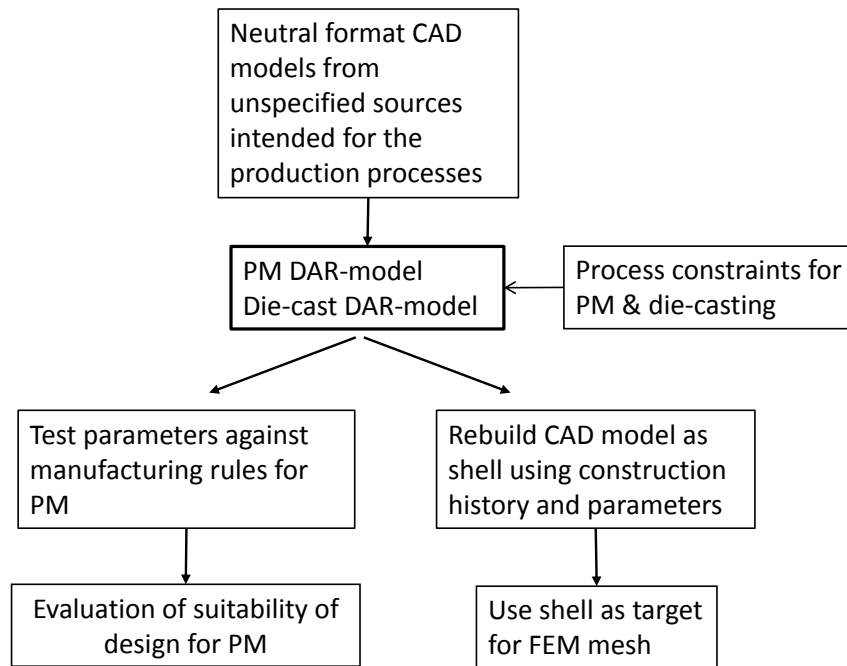


Figure 5.2 The use of DAR-models in the PM and die-casting context.

The types of DAR-models are different between the two processes. Nonetheless, they show similarities in that both methods are die-dependant. The intent is either to provide a basic evaluation of the models checking that a number of requirements are fulfilled, or to make more detailed predictions of the model behaviour using FEA. The proposed architecture allows the extension of the scope and precision of the system where the DAR-model can form an important starting point.

#### 5.4 SUMMARY OF THE RESULTS

The explored way of working, formatting the CAD model to a DAR-model preparing it for evaluation or DA is to the author's best of knowledge new. Previous IT-systems for securing manufacturability and automated idealisation have all been application specific and in many cases intended for interactive work. Further, previous work have not prepared for continued work. They are limited to the intended process. Unlike previous work, the DAR-models provide a parameterised model. The contribution of the thesis lies in combining an open user revisable KM environment with extensive geometrical processing of neutral file CAD-models. This allows the CAD integrated tools to be applied to any CAD-model intended for a certain production method not only the interactively defined model.

The intended use in the industrial context is that a basic form of CAD to DAR-model converter program is provided as an off-the-shelf software for each production process. The converter is complemented with DA or evaluation routines fulfilling general purpose needs. The software can be used precisely as it is, for example as a web-service allowing CAD-models to be uploaded and formatted to DAR-models. They can subsequently be evaluated and DA can be automatically performed on them. This will secure that the basic design considerations are adhered to. Alternatively, to make the system more elaborate, the people working closest to the application can be allowed to have an active role in the KM and the adaptation of the

system to specific products and processes, extending its use to include the management of design and production process knowledge related to particular products.

The advantage anticipated from separating the DAR-model from its subsequent DA or geometrical evaluation is flexibility. The DAR-model can possibly be used in different contexts and for various purposes, providing a modular structure of the program.



## CHAPTER 6

# APPLICATION EXAMPLES

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## CHAPTER OUTLINE

The work has been conducted through two application examples. A system for automated geometrical idealisation between the years 2003-2006 and a design advisor for PM parts 2006-2008. The experimental systems were partly programmed in the VB interface in CATIA V5, and the projects also included some stand-alone programmes coded in VB for Microsoft Windows XP™. The resulting experimental systems are accounted for in this chapter. First, the advisory aid for PM parts is described. Second, the die-casting example is presented.

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### 6.1 APPLICATION TO THE PM PROCESS

The design of PM parts is often done by the end users, i.e. the design engineers responsible for introducing PM components in assembled products. Designing the PM parts is often difficult due to the close relationship between the product and its production process. However, DFM issues inevitably must be taken into consideration since the parts' resulting properties largely depend on the design of the pressing tooling. Clearly, it would be useful to have an advisory system to aid the design process. Ideally, such a system should include the principal design of the pressing tooling. This is because there are probably several different alternatives for how to design the tooling for a particular shape of the part, all resulting in different mechanical properties. The system should therefore connect the tooling design with the resulting properties in such way that the part will display the best properties in regions where they are most needed.

Automated tooling design is complex. The whole pressing cycle must be considered from powder filling to, if needed, powder transfers, compaction and, finally, the ejection of the ready green part. Planning this cycle and designing the tooling elements require detailed knowledge about the pressing equipment (for example the stroke lengths of the different hydraulic actuators in the press). There are also a number of other decisions that have to be taken such as what tooling elements should form the different parts of the geometry. To simplify the design process, general rules and recommendations as mentioned in Chapter 3 can easily be used by the designer in that the tooling design does not have to be established first in order to use them. This is done at the expense of much lower precision since the rules have to be valid for all types of tooling configurations. It should be easy to implement these rules in an advisory system. This will be done to form a starting point for exploring automated advisory tools for PM parts design so that, from programming and operating the system, it will become clearer what other functions the system needs to have. In order to make a system capable of aiding in detailed planning and the prediction of properties in the part, tooling design will have to be taken into account. This is however, only briefly discussed in this thesis.

### 6.1.1 Defining the system

Provided that a design suggestion, represented as a neutral CAD-models exist, is should be automatically checked so that it complies with the geometrical design recommendations of the PM process. This will provide a “spell check” for the PM parts suggestion, checking that it has been reasonably designed. Using a rule-based system for this purpose should be a first hand choice since the knowledge is represented as a number of geometrical conditions which should be met. Therefore it is well suited for rule-based representation. First, the neutral file will have to be parameterised, making a DAR-model. It has previously been shown (in Chapter 3) how PM parts can be decomposed into manufacturing features. A parameterised version, with the manufacturing features now parameterised is automatically constructed in the CAD-system using its modelling commands. These commands are deployed by using VB scripting. Having a parameterised model will facilitate the creation of a rule-based evaluation of it, since purposeful parameters now are available.

### 6.1.2 Reconstruction process

As already concluded, one of the design requirements for the PM process is that the parts have to be designed in a limited number of levels for tooling design reasons. These levels can be retrieved from a CAD-model design suggestion by finding edges with coordinates in the pressing direction all located in the same or nearly the same levels. These edges are sorted out from the model and fitted with parameterised lines and arcs as described in Chapter 5. The lines and arcs in each level are thereafter assembled into closed loops. There can be one or several loops in every level. Further, the loops in the individual levels can be completely separated or contained in each other. For topology reasons, no loops can partly overlap each other since it was a valid model from start (see also Paper V). As a result, if intersecting loops were present it would not be a valid model. Once the loops have been reconstructed, it remains to reconstruct the solid model. This has been done by first extruding all identified loops through the full height of the part as seen in the sample part shown in Figure 6.1.

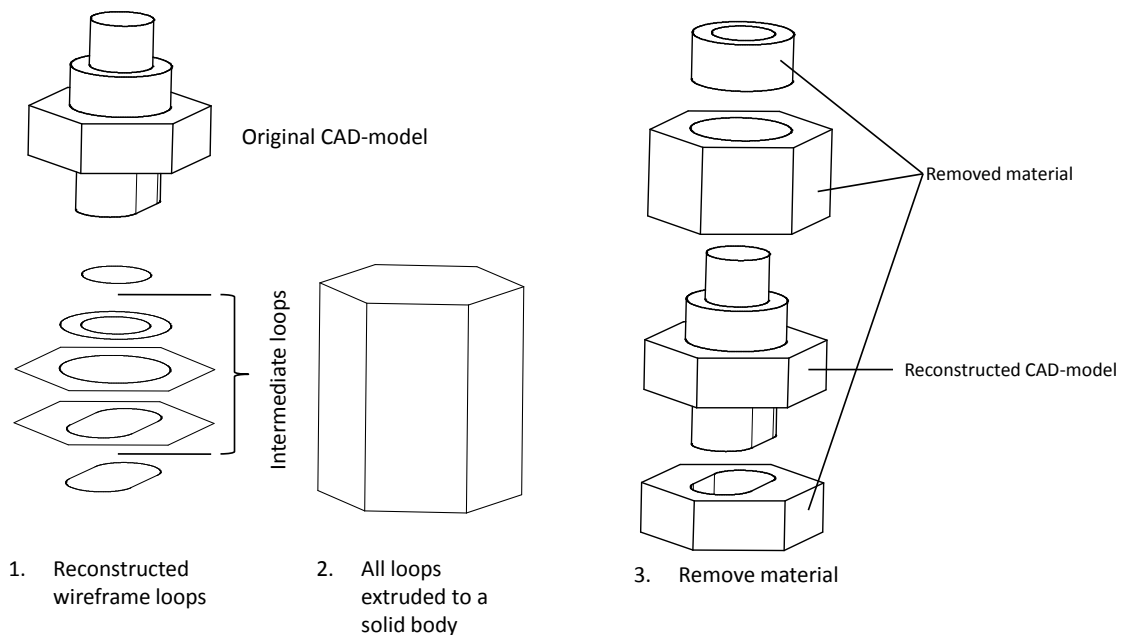


Figure 6.1 The solid-model reconstruction process (Illustration adapted from Paper V).

Secondly, the excess material must be removed using the intermediate loops as shown. To correctly reconstruct the solid-model, the side of the loop where the material is present must be known. Four cases (A-D) are then possible as depicted in Figure 6.2 below.

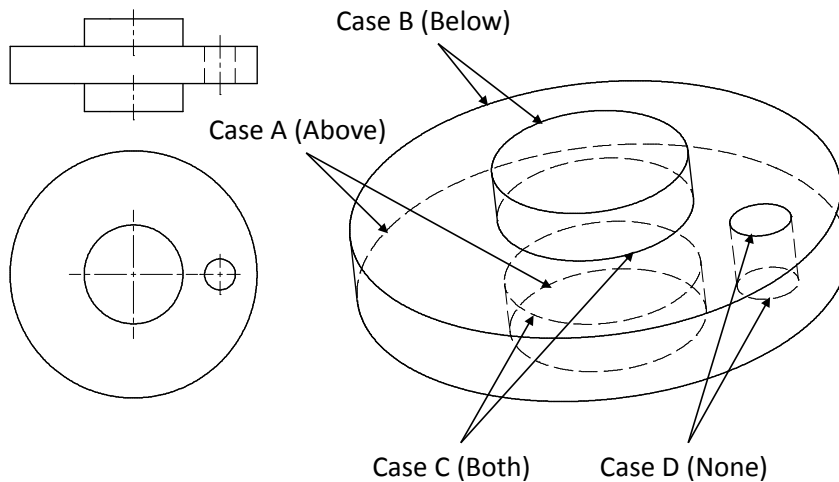


Figure 6.2 Location of material relative to the loop

In case D, there is no material present on neither side of the loop, i.e. a hole through the solid. There can be material present on both sides of the loop (case C). Finally, there can be material above or below the loop (case A and B). When reconstructing the solid, the extrusion direction of the feature to remove the excess material is the opposite of the material directions in cases A and B. In case D material is removed in both directions and no material is removed in case C. If the loops in a level are found to be contained in each other they must in some cases be groped in a single sketch to accurately reconstruct the solid. As depicted in Figure 6.3 there is in case 1 an outer loop that completely encloses the other loops. The outer loop can enclose a single loop as show in case 1a or several loops as in case 1b. In case 2a there is no outer enclosing loop. In case 2b there is an enclosing outer loop but the enclosed loops have material side none (case D). An important observation is that Case 1 can occur only when loops belonging to case C are present in the level. If the loops where not grouped into a single sketch extruding, the outer loop in its opposite material direction would remove the protruding features it encloses. If the loops are assembled in a single sketch, the solid-modeller will only remove the material located between the loops. For each level containing case C loops, the single sided loops are compared. The single sided loop with the largest radius is grouped with the case C loops and extruded in the opposite material direction of the single sided loop.

To summarise, the algorithm follows the below steps:

1. Among the intermediate level loops check the presence of loops with material direction both (case C).
2. In the levels found, find the largest single direction (case A or B) loop.
3. Assemble the case C loop and the largest single direction loop in one sketch and extrude it in the opposite material direction of the single sided loop.
4. Loops found at the ends (non intermediate) are extruded trough the whole solid provided they are case D loops.
5. Intermediate loops that are found in levels where none case C loops are found are extruded in their opposite material direction.

A special difficulty occurs when blind holes are present in the part. These will be wrongly interpreted as trough holes. This is because a case D loop is found at the entrance of the hole introducing a feature to remove all material through the solid. A check must be added to determine which type of hole it is.

The method used to determine which of the cases A-D a loop belongs to is by inserting a small feature that removes material. The feature is inserted in the interior of the loop, near the boundary. An assumption is made on which direction the material is located. If the volume of the part decreases when the material is removed then the assumption was correct and thus, the material side has been established. It remains however, to check that the sample location is not within the boundary of the enclosed loops (if any present).

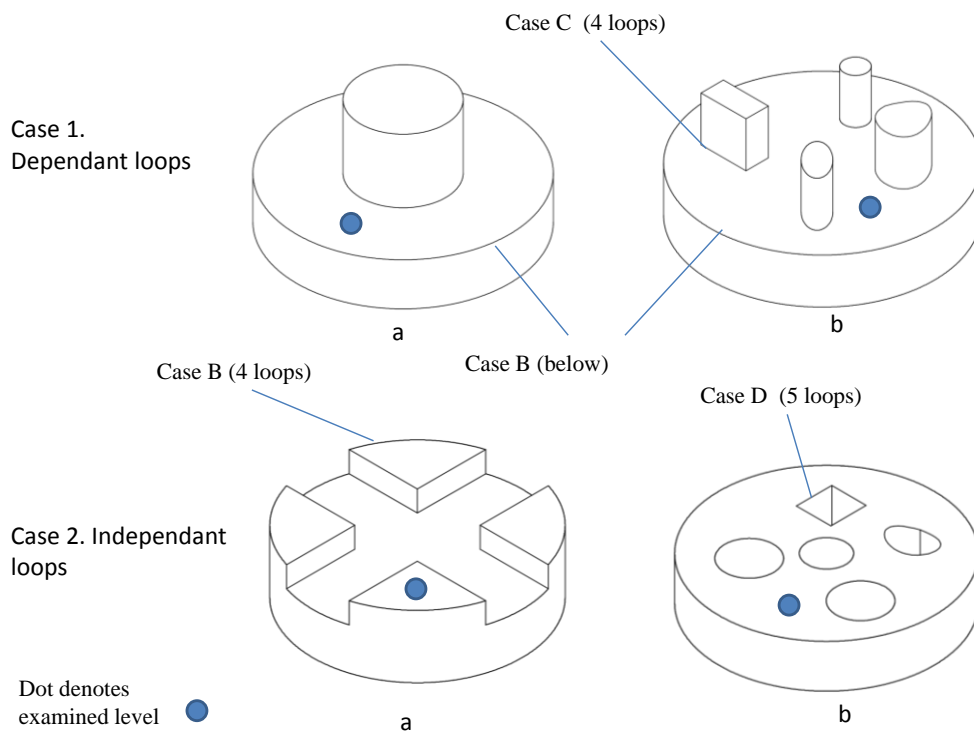


Figure 6.3 Dependant and independent loops.

The described reconstruction process address the primary features (powder columns). However, there are also the secondary features and face forms. These are not addressed by the current system. However, since they for pressing tooling reasons (levelling requirement) can be expected to be located near the levels and have a limited spacing in the pressing direction (due to the limited height requirement). The loops must be located near each other such as is show in Figure 6.4 in the next page for a chamfer and a face form. It should perhaps be possible to detect these features by finding the closely located loops. However, this has yet to be looked into. Further, there are also other considerations to be taken such as separating them from low powder columns. The type of face form feature could possibly be identified by its characteristic dimensions in the same way as the user features in the die-cast case.

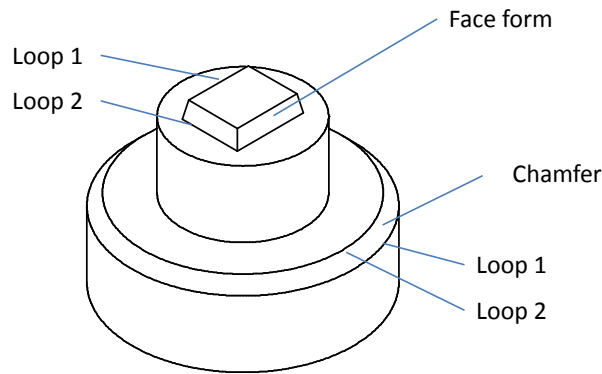


Figure 6.4 Identifying secondary features and face forms in a PM part.

## 6.2 EVALUATION

The parameters of the DAR-model are directly available for evaluation. The geometrical conditions that are checked are listed below. What the tooling implication would be if any of the geometrical conditions are found is also listed.

- Presences of narrow sections in the loops (difficult powder filling, fragile tooling elements).
- Presence of feathered edges (fragile edges on tooling elements).
- Small loop to loop distances (difficult powder filling, fragile tooling elements).
- Diameter height ratio (Low density or cracked part at ejection).
- Radii in corners (Powder flow needed for even filling).

All geometrical conditions are detected by IF statements. A value giving the threshold for when a certain condition is considered to having occurred is also given as user adjustable parameters. All except the diameter height ratio apply to the reconstructed wire-frame loops, which applies to the solid sections. The different conditions are depicted in Figure 6.5 below.

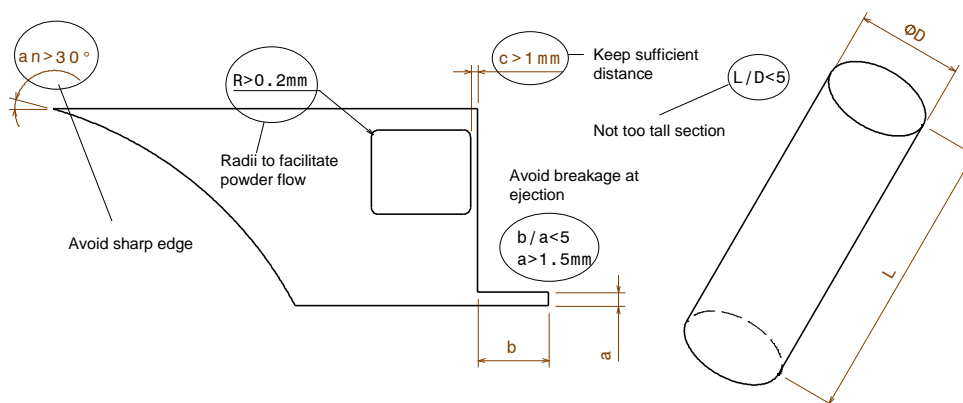


Figure 6.5. The different geometrical conditions checked. (Illustration from Paper IV)

The reconstructed sketches and their parameters can be used directly in the evaluation rules. In some cases, scripting has to be employed to derive secondary parameters from the model parameters as needed for the evaluation. A small angle between two elements in the loop is not recommended, since it will form a so-called feathered edge. It is a sharp edge in the part



that will wear off quickly, thereby reducing the tooling life. To reveal the presence of feathered edges, the angles between the elements in the reconstructed wire-frame are measured as seen in Figure 6.2. This produces a record of values for angles on which a rule-based evaluation can be made. Should one of the angles have a value less than a predetermined value, the affected rule is fired.

The reaction that follows should as much as possible inform the user of what type of geometrical condition that has been detected, where in the model it was found, what implications it has for the tooling and what can be done about it. This has not been particularly elaborated in the papers since the work has been directed towards creating a functioning system. The systems are not yet a state where user interaction issues can be particularly looked into. A message is displayed. Further, which rules were fired and which geometrical elements were affected is recorded. However, to arrive at a user friendly system, the graphical user interface (GUI) should be further developed. The location could for example be indicated graphically by a flashing red dot or similar. The message should also provide helpful information as to how to correct the detected problem. The user can then revise and re-run the geometry until no more rules are fired. This will ensure that the basic geometrical recommendations have been taken into account.

#### **6.2.1 Adjusting the detection parameters**

Representing the rules explicitly in the construction history tree of the model will mean that they can easily be accessed and revised by the users. The thresholds values to detect the various geometrical conditions can be altered. This could be done by running former designs in the systems. If any of the rules are fired when a well-working in-production part is run, then clearly the threshold value is wrongly set. The reason is that handbook recommendations have to include all materials and tooling technologies. They have consequently been selected with a large safety margin to accommodate all products and processes. By allowing adjustments to the threshold values several rule-bases could be derived, each adapted to the intended production process.

#### **6.2.2 Application examples**

The system has been developed and verified by running a number of sample parts in it. A selection of PM-parts that have been processed are shown in Table 6.1 on the next page. All the parts were found to be suitable for pressing, except for a number of feathered edges that were found in locations indicated by the arrows.

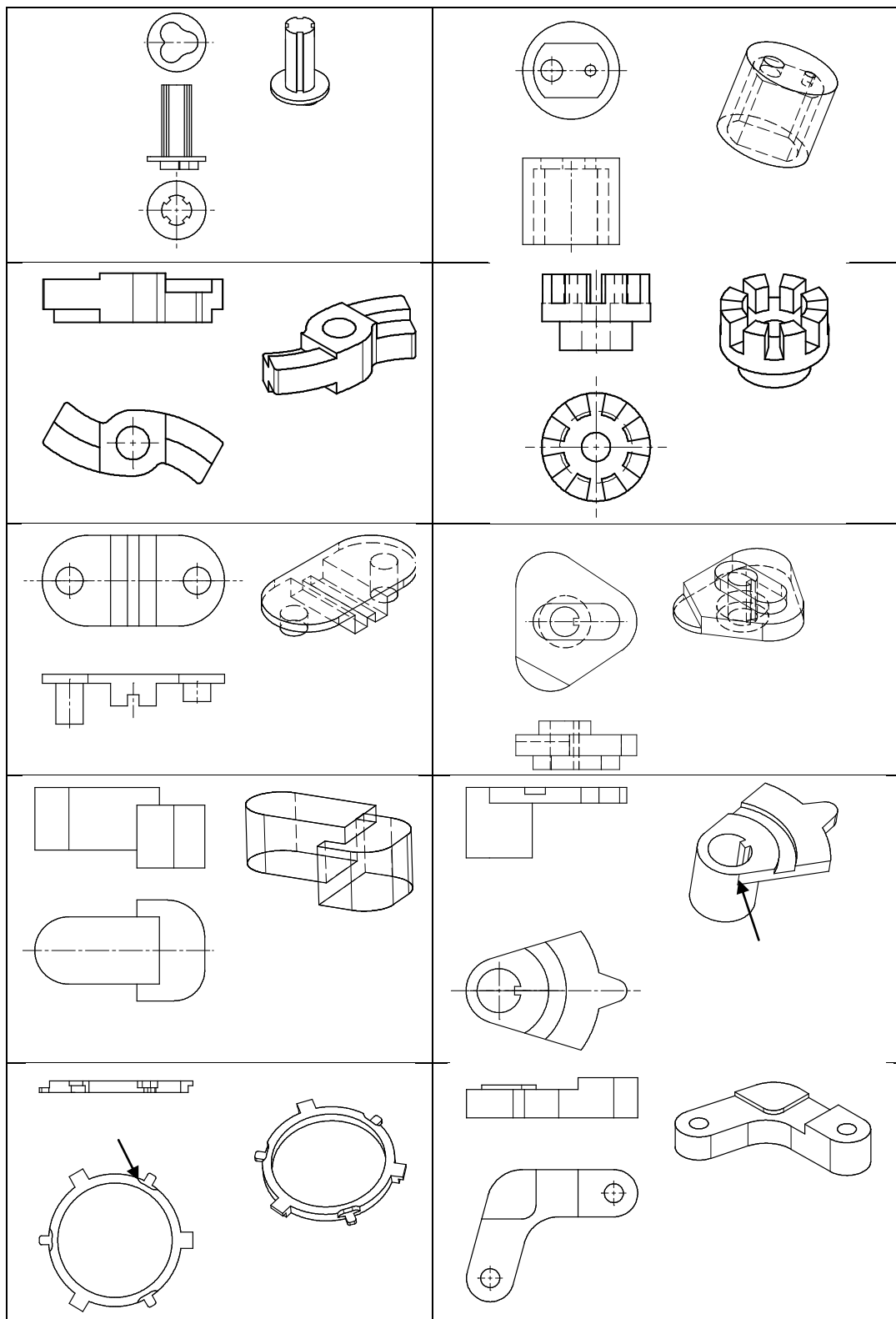


Table 6.1. PM-sample parts processed. (Illustration from Paper V).

### 6.2.3 Automated tooling design

When having the parameterised wireframe in tooling levels and their associated material direction these can be used to construct the tooling elements, instead of reconstructing the solid model. This is due to the fact that there must be a tooling element in the opposite direction of the part's material direction when the pressing cycle of the part is completed. This can form a starting point for the automated tooling design system. There are, however, many considerations that have to be taken into account when finalising the tooling design. First, the lengths of the tooling elements and how they should be actuated by the press have to be established. This and many other considerations are given in Paper V. However, the proposed reconstruction procedure could construct parameterised models of the tooling elements, so that time could be saved in the process of modelling them. Further, the created tooling elements could be analysed from various aspects.

## 6.3 APPLICATION TO THE DIE-CASTING PROCESS

The second application example involves the creation surfaces to be used as targets for shell element meshing. The need is partly motivated by the work-flow in the studied company (Paper I). The company develops die-cast parts to be used as components in the company's consumer products such as chain-saws, lawn-mowers and other foresting and gardening equipment. The parts are die-cast in-house, so the company has full control over the whole process from design to manufacture. The PD process starts by making a preliminary design. When doing so, the die-casting process is central. The actual part is not primarily the object of the design process, but rather the tooling to produce it. A CAD-model of the tooling dies are created, rather than the part itself. To derive a CAD-model of the part a Boolean operation is made creating an inverted version of the tooling cavity, representing the actual part.

The part must withstand the stresses it is subjected to in operation. In order to check this, FEA is employed. The FEA department then gets the solid-model of the preliminary product design and runs a number of load cases on it, verifying that the design can meet the structural requirements. In some cases, it is a clear advantage to have a shell element model. Indeed, for some of the calculations, shell elements are the only option. Otherwise, the calculation time would be too long. However, importantly, the advancements in computer power have meant that more solid geometry meshes can be used. When a shell element model is to be created, the solid-model is used as the starting point. For the thin-wall regions (such as the reinforcement ribs and the shell feature), the mid-surfaces are extracted. There are also areas of thicker geometry for which the mid-surface cannot be constructed. Instead, the FEA department has arrived at representations that successfully approximate the solid regions for the analysis conducted. The mid-surfaces and the surface representations of the thick regions are then trimmed together to a complete connected target surface of the model. Finally, the target surface is meshed using shell elements.

The process of transferring the solid-model into its surface representation is done interactively in the CAD or FEA program. It can be seen as a repetitive design task involving both general and product specific knowledge. To save time in the process and ensure the quality of the resulting target surface, the task could be partly of fully automated using the presented technologies. This would enable storing and automatically reusing the knowledge of how to make the surface representation.

First, the features in the CAD-model need to be identified. Second, automated routines have to be defined to automatically insert the desired surface representations. This must be done in a way that enables the user management of the knowledge.

### 6.3.1 Finding the features

In Papers I and II, a prerequisite for the described system to work is that the parts have been parametrically constructed in the CAD-system. This has to be done by first creating sketches and then linearly extruding them in the tooling draft direction. The features' sketches and the extrusion lengths are then used to automatically create mid-surface representations by constructing the medial segments in the sketches and then extruding them. However, the proposal of requiring that the complex die-cast geometries should be modelled using only linearly extruded geometries in a very restricted way will reduce the usefulness of the CAD-system to a large extent. As such, it is clear that it was not a feasible way for the system to work. Consequently, in Paper III this problem is addressed by proposing interactive identification of features in neutral file die-cast geometries. When the features have been interactively identified, appropriate surface representations are retrieved and inserted in the correct place in the model.

Having to identify the features interactively will add to the time spent on creating the surface representation. It will also negate the possibility of running the system in automated knowledge acquisition processes or optimization loops, for example. However, the method that was developed when working with the PM parts application example (There the edges of the model were retrieved and parameterised) can also be used on the die-cast geometries. This is more complicated since the die-cast part does not necessarily have the characteristic levelling of the PM parts. The rib and user features can start in any surface and extend up to the shell feature or end. However, it should be possible to find the features by projecting the model's edges on planes perpendicular to the tool opening direction using the method of retrieving the model edges and inserting parameterised wireframe entities. To make the projection, the z coordinates of the points are replaced by a constant representing the location of the projection plane in the z-level. In this projected set of wireframe, rib and user features can be sorted out using the same procedures as described in Papers I and II (where a rib feature is identified by finding geometrical elements that run in parallel approximately one material thickness apart). These are then paired, forming a thin-walled entity. User features are identified by characteristic dimensions and patterns of geometrical entities as described in Paper VI. All features, except the shell feature which have to be interactively identified, are found in this way. Still, the geometry may contain unidentified features. These may originate from feature interactions resulting in 2D geometry that cannot be resolved.

### 6.3.2 Constructing a surface representation

The object of the reconstruction process is to create a surface representation of the part, rather than a parameterised copy of the geometry, as in the PM-part case. Therefore, the paired segments are used to construct the medial segment between them by inserting the same type of element in the centre. This will result in an unconnected set of individual segments. A trimming algorithm to close the gaps that appear between the medial-segment has been defined (Paper I). Its working principle is to find the nearest intersection and extend the individual segments up to the intersection. The trimming algorithm has been further explored in the Bachelor's Degree work of (Tapankov 2007). Here, Tapankov found ambiguous cases in which the proposed trimming procedure does not suffice. Meta rule are required to select among the ambiguous trimming possibilities.

The trimmed wire-frame is then extruded through the original solid-model. The Figure 6.6 below shows a simple part for which the medial segments found in the projected set of wire-frame has been reconstructed and subsequently extruded.

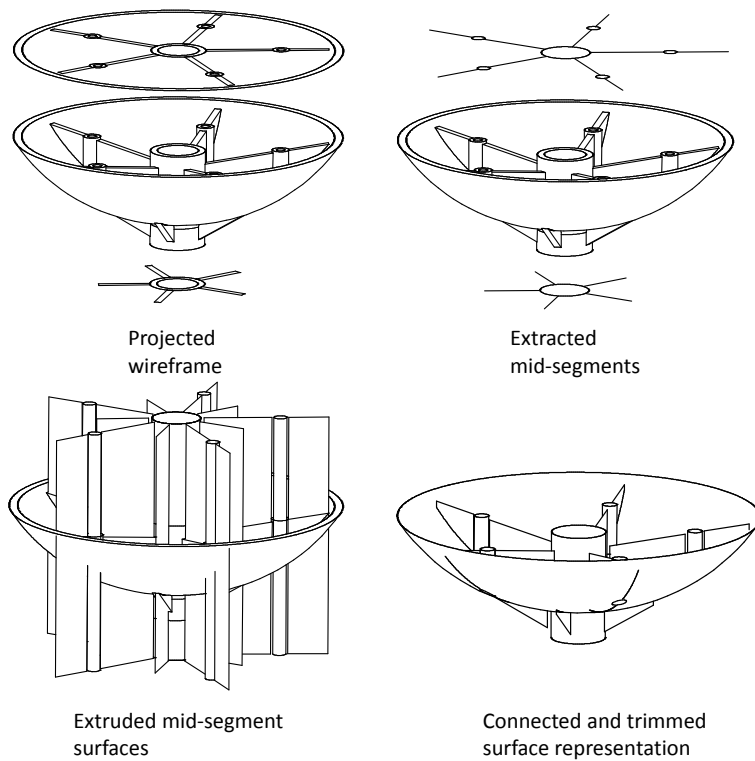


Figure 6.6 Creating a surface representation for shell element meshing.

The solid-model and the extruded surface can be intersected, leaving the portion of the surface which is common between the two entities (as described in Paper VI). However, it will result in narrow slivers (half material thickness in width) of unwanted surfaces in places where rib features intersect each other. Alternatively, the excess surfaces will have to be removed interactively by using the original model's bounding faces as trimming elements. The removal of excess surfaces was addressed in Paper II. However it is then required that simple modelling procedures are used i.e. extruded cuts perpendicular to the draft direction. This will allow the features to be detected, so that a corresponding piece of the surface representation can be removed. It was found that the die-cast geometries could not be modelled using only these simple commands. Consequently, this problem remains unsolved and will have to be addressed interactively.

## 6.4 MANAGING THE KNOWLEDGE

In the experimental system for idealising die-cast geometries the working principal is mostly procedural. However, rules have been employed in some cases to trigger these procedures. The knowledge needed to create the idealisation cannot be successfully represented as simple rules, as was the case with the PM-parts. Extensive processing is needed to derive the medial-segments of the projected wire frame and to insert the appropriate representation. In both the die-cast and the PM case, to allow active KM, the knowledge must be represented in a way that can be read and revised by the user to adapt it to local circumstances concerning the products and processes in the particular case. As seen in Figure 6.5 the knowledge needed can be represented in a template CAD-file which can be imported to the file containing the DAR-model. The knowledge can subsequently be applied to the DAR-models features and parameters. The different types of knowledge needed are represented as "knowledge items" in

the construction history tree of the template CAD-file. The sections of the user features and possibly the face forms can be defined in prototype feature sketches, so that when making the FR procedure in the neutral files, these features can be identified. This is done prior to converting the model to DAR-format. Also the rules, scripts and the systems constants needed can be represented in the template CAD-file. Examples of system constants are thresholds for determining the thickness of a thin-walled feature and determining how close in the pressing direction the points must be located to be considered to belong to the same loop.

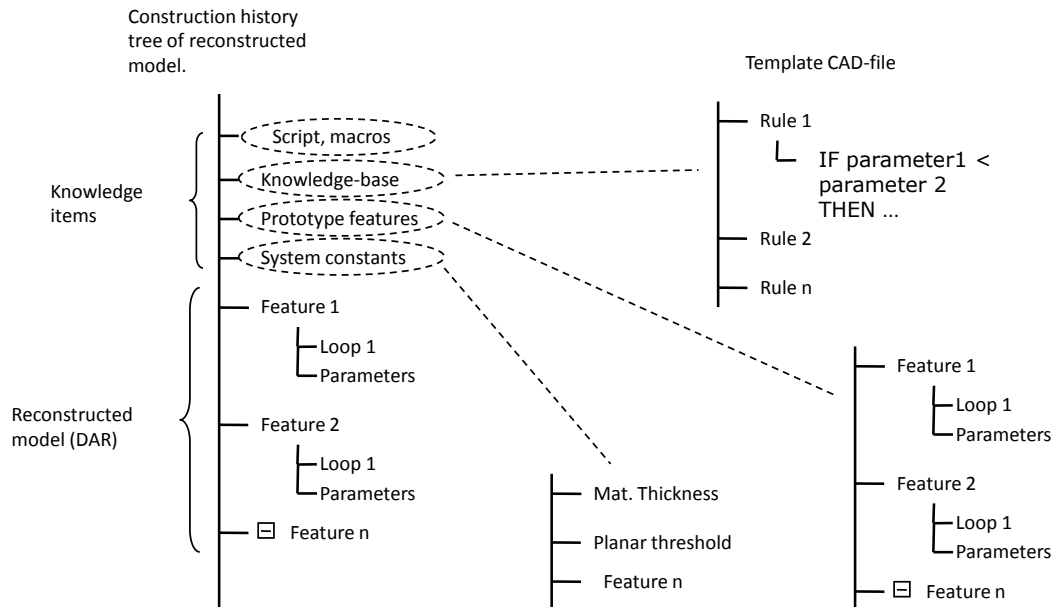


Figure 6.5 Managing knowledge items in template CAD-files. (Illustration from Paper VI).

Storing the knowledge needed in this way will make it directly applicable to the parameters and construction history of the DAR-model and also accessible and revisable to the system user.

## 6.5 PROGRAMMING ENVIRONMENT

The programmed procedures in CATA V5 showed extensive similarities and several modules from the first example could be reused in the second and vice versa.

The programming examples provided in the CATIA manual (Dassault-Systemes 2008) were, at least when the work was carried out, quite sparse. There have been several modules developed to conduct the application examples, all with different geometry creation and manipulation functions. For the interested reader who may be pursuing automation projects and coding in CATIA V5 some samples of source code is provided in the appendix of this thesis. It is essential for the construction of the DAR-models that the coordinates of each point on the model edges can be read so a simplified function showing how it has been done is therefore provided. The function "GetAllEdges" retrieves all edges in the model and "Add3PointsOnEdge" shows how the start, mid and endpoints of the edge is fitted with points and the global coordinates of them returned. It is also shown how the material side of the loops can be categorised using an automation of the CAD-systems measurement tool.



## CHAPTER 7

# CONCLUSIONS AND FUTURE WORK

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## CHAPTER OUTLINE

This chapter summarises the conclusions and experiences made when conducting the work. It answers the research questions, and it proposes how the work can proceed in the future.

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It is possible to define a parameterisation and construction history for CAD-models intended for manufacture by the PM pressing and sintering and the die-casting processes. It has been shown how to automatically derive it from CAD-models of any format. This structure will enable further automated design and design evaluation of the CAD-models. This means that the functionality of the CAD system can be used on all CAD-models intended for the processes, regardless of their origin. These models, called DAR-models here, may not be practical to be created by interactive modelling. As such, an important conclusion is that a FR, rather than a design-by-features approach must be adopted for constructing them. The FR approach also means that the DAR-models can be used in optimisation loops and automated knowledge acquisition in that the process can be automated.

The FR approach is based on the principle that the features produced by the production process can be categorised as primary, secondary and specialised. For primary and secondary features, software to handle all CAD-models can be defined for such a task, since these features have similar shapes in all parts produced by the processes. The specialised features, however, cannot be handled by standardised software. Even if they follow restrictions given by the production processes, their exact shapes depend on their intended purpose in the products. Consequently, the FR approach system must be complemented by the means to allow the user to define the shape of the specialised features. Subsequently, the FR system must be able to identify them and the actions that should follow once identified must also be specified. A way of accomplishing this using template CAD-files has been explored.

Since there is a clear distinction between on one hand the primary and secondary features and on the other hand the specialised features, the software must be designed so that it provides one standardised section for identification and processing the primary and secondary features. It also needs to provide an environment that allows the user to extend and adapt the system to the particular application. The software solution can operate on two different levels. First, it can be used as a check to secure that the basic processing prerequisites have been adhered to when establishing the parts geometry. Since the knowledge needed to do so is not expected to change unless the working principle of manufacturing process is changed, it can be provided as an off-the-shelf software solution ready to be distributed and deployed anywhere where products sharing the same manufacturing processes are being designed. Second, to make a more detailed evaluation of the suggested design, the users must be able to extend the system and adapt it to the specific products.



## 7.1 PRODUCTION PROCESSES

An important reflection when developing the systems is that the proposed working principal seems to work better the more shape restrictions the process imposes on the geometry. The automated idealisation of the die-cast parts only allowed itself to be partly automated. It was, for example, not possible to find a way of identifying and idealising the shell feature. Further, intersecting the extruded surfaces with the original solid did not result in the desired target surface. Consequently, much is left to interactive work. For the PM-parts however, with their more elaborate shape restrictions, it was possible to, albeit disregarding secondary features, automate the reconstruction process fully.

To conclude, the die-cast system should act as a support tool for the interactive work on the parts. One way is by constructing parameterised wireframe and extruded surfaces, leaving the trimming to interactive work. This will reduce the time spent on making the idealisation. The PM-application, on the other hand, is a more likely candidate for full automation. Consequently, future work should be directed towards developing the PM-system toward full automation.

It has also been found that a complete ready to use system including all product and process knowledge can be defined neither in the PM nor in the die-casting case. In both cases, the systems have to provide possibility for user adaptation to the specific products.

## 7.2 CONCLUSIONS RELATED TO THE RESEARCH QUESTIONS

The conclusions related to the research questions are summarized in this section. The research questions from Section 1.5 are first repeated.

**How can a preliminary geometrical specification of a product together with production process constraints be used as a starting point for further automated design and evaluation of the product?**

The explored way of accomplishing this for preliminary designs intended for the die-casting and PM pressing and sintering processes is by employing a FR approach that is based on the shape restrictions of the production processes. This allows the automated identification of a limited number of manufacturing features. The manufacturing features are typical shapes produced by the process. These can be automatically reconstructed in the target CAD-system by using programmed scripts to create CAD features corresponding to the identified manufacturing features. This will result in a CAD-model with predefined construction history and parameters (DAR-model), enabling the direct utilisation of the CAD-systems KM and reuse functionality.

**How can the need of transparency and the maintainability of product and process knowledge be addressed when exploring the first research question?**

The intent behind the CAD-systems KM and reuse functionality is that the user should be able to use them without being an expert programmer and work in a familiar environment. The functionality is primarily intended for use on the interactively defined model. However, the DAR-model allows this functionality to be applied to any CAD-model. In this way a transparent and directly revisable KM environment can be established for use on any model. This can typically be managed by a group of engineers working with the particular type of products enabling them an active role in the KM and reuse of engineering knowledge.

### How can the above support the development of new products?

The systems can be used for mainly two purposes: for the active knowledge management and automated reuse in the organisation and to share the information externally. The external information sharing is realised by accepting any CAD-model design suggestion and applying the gathered process and product knowledge to it. In the PM-case former design can be converted to DAR-models allowing them to be evaluated in the system so that a build-up and refinement of knowledge can be conducted.

## 7.3 VERIFICATION

The work presented is primarily based on a number of sample parts. These parts have been obtained through the companies that have participated in the two research projects. The parts have been carefully selected, and include parts that currently or previously have been in production. Care must be taken when selecting the sample parts so that they truly are representative of the studied problems. Without the important participation from industry, there is a risk of inventing sample part that are overly simplified and do not represent the real problems. In addition to these, in the PM case, the sample parts have been complemented by additional parts found in handbooks. The work has been based on approximately 10 die cast parts and 30 or more PM-parts. The die-cast parts are very complex so simplified versions have been created as needed. The PM parts' geometries could be kept as is, yet with secondary features removed. All parts were idealised, meaning the secondary features were removed. Since the conclusions are drawn from experiences made when developing and operating the experimental systems, it is important to verify that the system work as intended. This is done by running a number of sample problems in them. If the results are the expected, the system is considered verified. Verification is not only made on the complete system. During its development, all programmed procedures and modules were verified. If only the complete system is tested and the results are erroneous, then it is will probably be very difficult to find which component or components are responsible. Therefore, every function in the system is verified by a set of sample problems. The functions presented in the appendix have been verified in this way.

This method of verification does not guarantee accurate results for all parts produced by the process. Situations not anticipated can be encountered. Nonetheless, since the parts were selected from the companies' part portfolio, the risk is reduced.

## 7.4 VALIDATION

Validation means ascertaining what extent and under which circumstances the results are valid. In the context of this thesis, it means finding the significance and industrial impact of the results. If the problems studied generally exist, then the solutions to them also are applicable in all similar situations and not only in the cases studied. Two circumstances indicate that the problems studied generally exist. First, the companies have been extensively seeking solutions to the problems themselves. If this is done for a period of time involving several people and the problems seem to defy all solution attempts, then this clearly suggest the presence of a knowledge gap. Second, if the researchers also fail in finding an existing solution to the problem, it will further ensure the relevance of the problem. To close the knowledge gap, a research project is then well motivated.

## 7.5 CONTRIBUTIONS

An important contribution of this work is the introduction of the DAR models. These allow a modularisation of the system, so that the reconstruction procedure can be conducted and different applications can follow building on the same parameter and features structure. It has been found that there is a generally applicable knowledge related to the basics of the production process and detailed knowledge related to the specific product and processes. The consequence is that, in order to elaborate the system, it must be adapted to the local circumstances.

It has been shown that CAD-integrated means of managing and automatically re-applying knowledge is useful to provide the user manageability and adaptation needed. The work has also provided a categorisation of production features into primary and secondary, which are generally appearing features in die-cast and PM parts, and specialised features. This provides a partition into features that can be generally handled and those that require adaptation.

### 7.5.1 Scientific contribution

The research process has followed the DRM introduced in Chapter 2. It emphasises the importance of establishing measurable criteria so that the improvements can be assessed. Here the participation of the companies has been important. It has not yet been possible to conduct any comparative studies, comparing the suggested tools to alternative methods, due to the immaturity of the systems. However, they have been presented to the organisations beginning to identify the important factors and ensuring that the demonstrators are focused on the issues that they are intended to address. The following step should therefore be to proceed from working with sample parts into starting to involve working professionals. This would possibly lead to that more such factors can be identified leading to an increased refinement of the knowledge about the suggested type of systems. The current situation is that a number of sample parts have been run with the expected results in the systems. The participating companies have helped to assure that the selected samples are representative. The systems can therefore, at the time being, be said to have been more thoroughly verified than they have been validated.

### 7.5.2 Industrial contribution

The PD process was discussed in chapter 1. It was recognised that in order to reduce the time spent in the design loop one possibility is to reduce the time spent to model the envisioned design for subsequent analysis. The suggested systems can help to do so at the expense of building and maintaining the system. Identifying which part of the knowledge is generally applicable and which requires adaptation is therefore an important contribution. One can start with off-the-shelf software and if it turns out to be successful gradually expand involving also the company specific knowledge.

Geometric advisory systems to enhance manufacturability are, as shown, nothing new. However, combining geometric processing with open CAD integrated tools for KM and reuse is, at least for single die-cast and PM part not previously tried.

The necessity of adaptability of automated idealisation to specific product and processes has not previously been recognised to any large extent. The tradition of the software companies is to try to anticipate, possibly by surveys, the user's needs. When a need has been identified it is typically addressed by adding new functionality in the software. Starting to investigate how the users themselves can address the problems by using the CAD-systems facilities is an important step in this development, which previously have not been addressed.

## 7.6 FUTURE WORK

Ultimately, the PM application holds the promise for full automation. However in the die-casting case, it seems to be very complicated to fully automate the FR process and the complete automated construction of the target surfaces. Therefore, the work there should instead be directed to supporting the interactive work.

The secondary features have currently been disregarded. Some of them, such as outer edge chamfers on the PM part, are undoubtedly important for the design of the part. Developing FR methods of finding them, perhaps in by finding adjacent loops in the way suggested in Section 6.1.2, is important. Thus, it should be the first priority of future work. The effect of the secondary features could then be taken into account, and parts could be processed without having to be de-featured first. Further a procedure to determine if a point is located inside or outside the boundary of a loop is needed to accurately classify the type of loop and to determine if a hole is blind or through.

The method implies that the design suggestions are quite well adapted to the process, allowing accurate identification of the features. The actions the systems take when a part with unidentifiable features is entered should be looked into.

Currently only handbook recommendations are evaluated. In order to provide more precise advice and to establish a connection between the parts' geometry, its requirements and the resulting properties, it is necessary to involve the tooling design, the process route and the material. Preliminary tooling elements can be obtained as demonstrated from the DAR-models. However, to finalize the tooling, the pressing equipment has to be known. The pressing tooling should then be planned in such way that the requirements on the part direct the design of the tooling.



## CHAPTER 8

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# APPENDED PAPERS

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The appended papers have been formatted to suit the layout of this thesis. Minor changes have been made to clarify figures, spelling and formulation. When referencing to the author's own papers the original references have been replaced by Paper I-V.

## Paper I

Stolt, R. and Sunnersjö, S. (2005), "Automatic Preparation of CAD-generated Solid Geometry for FE-meshing", Proceedings of NAFEMS World Congress 2005, May 17-21 St. Julians, Malta.

## PAPER II

Stolt, R. (2005), "A CAD-Integrated System for Automated Idealization of CAD-Models for Finite Element Analysis", Proceedings of DETC2005: ASME International Design Engineering Technical Conferences and Computers and Information in Engineering Conferences -31st Design Automation Conference, September 24-28 Long Beach, CA, United States.

## Paper III

Stolt, R. (2006), "Reusing CAD-models of Die-cast Products for FEA", Proceedings of 2nd NAFEMS Seminar: Prediction and Modelling of failure using FEA, May 31 - June 1 Copenhagen / Roskilde, Denmark.

## Paper IV

Stolt, R. (2008), "Identifying Features in CAD-Models for Powder Metallurgy Component Evaluation.", Proceedings of Design 2008: The 10th International design conference, May 19-22, Dubrovnik, Croatia.

## Paper V

Stolt, R. (2008), "An Automated Design and Evaluation System for Pressed and Sintered Powder Metallurgy Parts", Proceedings of MPIF/APMI 2008 World Congress on Powder Metallurgy and Particulate Materials, June 8-12 Washington D.C, United States.

## Paper VI

Stolt, R. (2009), "Managing Product and Process Knowledge in CAD-Systems", Accepted for publication in the proceedings of NordPLM'09: The 2nd Nordic Conference on Product Lifecycle Management, January 28-29 Goteborg, Sweden.



# AUTOMATIC PREPARATION OF CAD-GENERATED SOLID GEOMETRY FOR FE-MESHING

ROLAND STOLT AND STAFFAN SUNNERSJÖ

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## ABSTRACT

In recent years, computing power and meshing algorithms have developed to a state where FEA problems can often be solved directly using solid geometry. However, for complex geometry and complicated calculations, there will be a need for geometrical idealisations for the foreseeable future.

To reduce the time spent on making geometrical idealisations in repetitive FEA, a CAD-integrated KBES (Knowledge Based Engineering System) has been developed. The KBES creates a surface idealisation from a thin-walled solid by utilising generic modelling knowledge and by registering information about the CAD-specific features, which the designer uses to define the solid geometry. From this information, a corresponding surface idealization is created in the same CAD-system. This allows an updated and parametric geometry idealisation of the complete CAD-geometry to be created with a degree of automation directly in the CAD-system.

Evaluating the sketches that the features of the CAD-model are based on creates, primarily, the mid-surfaces oriented in the tooling draft direction. The KBES also trims the created surfaces, thus facilitating the subsequent meshing.

The KBES has been developed in CATIA V5 (Dassault-Systemes 2008). It contains rules defined in CATIA knowledgeware that trigger sequential routines written in VBA (Visual Basic for Applications). An industrial application example is also presented in which the system is used to automatically create a surface idealisation for a die-cast part.

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## 1. INTRODUCTION

FEA often requires a geometrical idealisation of the real problem. In general, the process of making a FE-model based on CAD-generated data follows the following steps:

First, the analyst seeks to simplify the geometry, removing features of no interest. This is known as global idealisation (Dabke, Prabhakar et al. 1994). Second, the analyst tries to make a geometrical reduction of the problem. If the geometry is thin-walled, the solid may be replaced by a shell representation without undue effect upon model behaviour. Symmetry can be used for further geometrical reduction. This is called element idealisation. The third step is to choose

appropriate element types and to mesh the model. The loads and boundary conditions are also applied in this step.

This paper focused on the first two steps; namely, the global and the element idealisation of the geometry. An appropriate remark is that, in recent years, both the meshing algorithms and the computing power have developed very quickly. The time spent on geometric idealisations has therefore decreased significantly. On the other hand, increased demands on accuracy and more sophisticated analysis procedures including effects of complex physical phenomena multiply the required computing power. Despite the increases in computer power, it seems as though, for many computational problems, idealisation of some sort will be a necessity for the foreseeable future when carrying out FEA.

The process of manually idealising the CAD-data is time-consuming in many cases. Where possible, the idealisation should be generated automatically to free the analyst from this tedious work. Attempts to automate the idealisation step have been going on for at least three decades and have basically followed two different tracks, according to (Tworzydło and Oden 1993):

- A generally applicable theory that requires large scale algorithmic computations.
- A domain specific approach that utilises prior knowledge of modelling methods and geometry characteristics.

The first method has proved to be difficult. This is because it must be capable of handling many types of idealisation problems. Research is being conducted at Queens University of Belfast, for example (QUB 2004). The object is to use the MAT (Medial Axis Transform) of CAD-data to automate geometry idealisation both globally and for element reduction. In (Lee, Price et al. 2003), an example is presented where the MAT is used to secure that no unwanted topology exists in the CAD-data, thus facilitating the auto-meshing. How the geometry can be prepared for FEA by automatically suppressing details and reducing dimensionality is shown in (Armstrong, Donaghy et al. 1996).

To formulate an algorithmic procedure capable of handling all types of idealisation problems seems very difficult, since prior knowledge about the object and its design factors are not utilised.

The second method, according to (Tworzydło and Oden 1993), is restricted to a certain domain of problems. It assumes that CAD geometry is created according to certain principles and has characteristic features, due to the way in which the product is designed to be manufactured. Knowledge about the product helps to define the problem in a representative way. In these cases, a well-known strategy is to use KBE (Knowledge Based Engineering). The term KBE is a subset of KBS (Knowledge Based Systems). This distinction has been made to emphasise the geometrical side (Sandberg 2003). The term KBES (Knowledge Based Engineering System) refers to a system which utilises the KBE concept.

In (Prabhakar and Sheppard 1992), a KBS for geometry idealisation called DESIDE-X is described. In (Li and Qiao 2003), a KBS using a neural network to compensate for the shortcomings of traditional KBS is presented. Many of the KBS described in literature focus on steps following geometry idealisation. One example is in (Dolsak 2002), where a KBS is used to suggest element types and to determine the mesh resolution values. Here, the knowledge is gathered from ML (Machine Learning)

It is common that a facility for creating CAD-geometry is available in commercial KBES. Alternatively, some CAD-systems have been equipped with KBE capabilities. The purpose is usually to keep the product knowledge of the design at hand stored and executable to define variants of the product. An example of a KBES used to automatically generate an FE-model is described in (La Rocca, Krakters et al. 2003). Here, geometrical as well as engineering knowledge for a structural analysis of an aircraft design is provided by the KBES for evaluation in FEA. The

commercial KBES used is ICAD from KTI (KTI 2004) Another example exists in (Chapman and Pinfold 2001) where an automotive BIW (Body in white) is designed using AML from (Technosoft 2004). Here, the KBES provides an analysis model, runs the analysis and brings the results back into the KBES. In (Isaksson 2003), a KBES used for designing aircraft engines is presented. It is used both for generating and evaluating the designs.

## 2. AIM OF RESEARCH

The three KBES described in (La Rocca, Krakers et al. 2003), (Isaksson 2003) and (Chapman and Pinfold 2001) obtain an FE-model by letting the designer chose among engineering objects such as canards, struts, and pillars. The FE-models are adapted to the design situation. Consequently, these types of systems can only deal with variants of a similar product.

The purpose of this research is to analyse the features in the CAD-model, allowing a more general function of the KBES. The analysis is made under restrictions given by CAD modelling methods and the chosen manufacturing method, which in this case is die-casting. The aim is to build a prototype KBES with this working principle and evaluate its usefulness in an industrial case.

To the authors' knowledge, the approach with a KBES, which creates surface geometry for FE-purposes by analysing the CAD-features, has not previously been tried.

## 3. CASE STUDIED IN RESEARCH

The methodology described in this paper is tested on an industrial problem relating to equipment for lawnmowers, trimmers, and chainsaws. Most of the parts in these products are manufactured through die-casting. When designing a new part, the traditional way has been to first make a design proposal in the CAD-system. The solid geometry is idealised within the same CAD-system. Often, in the case studied, an external consultant makes the geometrical idealisation, since neither the design department nor the engineering department has the time to make the idealisation. The idealisation is exported to a FE-program and a FE-model is built and evaluated. If the result of the evaluation proves satisfactory, the design is handed over to downstream activities like tooling. The workflow is shown in Figure 1 below.

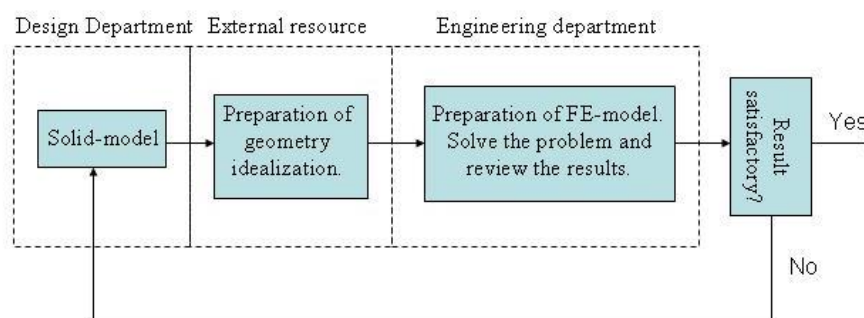


Figure 1: Traditional FEA workflow in the studied example.

The main objective of the FEA is to find out how the assembled product will behave under the influence of vibratory excitation from the engine. FE-models of all parts belonging to the product are defined. The individual parts are assembled into a FE-model of the complete problem. The FE-model is subjected to loads from recorded frequency spectra, and the response of the structure is calculated. The risk of premature failure is evaluated from the result.



The geometry idealisation is achieved mainly by replacing the solid geometry with the mid-surfaces. These can later be meshed using shell elements. The creation of the idealisation is considered unacceptably time consuming, often two to three weeks. To shorten the lead time and to make the results available at an earlier project stage, this work suggests a KBES, which allows the idealisation to be created with a degree of automation.

#### 4. METHODS OF CREATING SURFACE IDEALIZATIONS

As previously discussed, the commonly used methods for the creation of surface idealisation with full or a degree of automation are the algorithmic and the KBE approach. The former approach is quite general, and the latter is much specialised. The approach described here seeks to place itself between the two in terms of generality.

##### 4.1 General algorithms

General algorithms for mid-surface extraction exist. In (Rezayat 1996), an approach based on mid-surface abstraction is presented.

For the real-life components in this application, it has not been possible to produce the mid-surface idealisations using the automatic mid-surface extraction functions provided by the CAD and FE-systems. It has been reported that they can be used on simple geometry. However, they seem to fail as the complexity of the CAD-data increases. Possible reasons are that the mid-surfaces were not defined in some areas, or that the models contained errors the software could not deal with.

##### 4.2 Interpretation of CAD-commands

When planning the layout of the KBES, one of the first ideas was to use a commercial program to extract the mid-surfaces and afterwards let the KBES fill in or modify the created geometry in areas where other representations than the mid-surface were wanted. However, it was realised during the work that three major advantages could be drawn from creating a fully parametric surface idealisation of the solid geometry simultaneously with the CAD-model. The first advantage is that the idealization could be accessed at all times to be used for FEA. Second, the geometry idealisation is easily changed due to its parametric definition. Third, the designer can check that the created geometry is correct and change it if necessary.

Due to the nature of the manufacturing process, several assumptions regarding the geometry can be made. These assumptions will also largely facilitate the creation of the KBES:

- All geometry is thin-walled. The thickness is assumed constant in a single wall and roughly the same in all walls.
- Only one tooling draft direction is allowed.
- The outer appearance of the visible part is defined by a styling surface.

Real life die-casting tools are often more advanced and include several tooling directions. In this prototype KBES, only one tooling direction is included

#### 5. DESCRIPTION OF THE FUNCTION OF THE KBES

In order to test the feasibility of the ideas, a prototype KBES utilising the idea of interpreting the standard CAD-commands is built as described in the following sections.

### 5.1 Limitations of the KBES

In order to ease the creation of the KBES, restrictions were placed on what modelling techniques are allowed in CATIA V5.

- The model has to be built using only the commands “Pad” and “Pocket”. These are methods in CATIA V5 for adding and removing material by means of linear extrusions. Note that no draft angles are allowed.
- The sketches may only contain lines and arcs.
- All sketching planes must be placed perpendicular to the tooling direction.

### 5.2 Programming

The KBES is comprised of declarative rules that have been defined in CATIA V5 knowledge expert and scripts written in VBA (Visual Basic for Applications). The task of the declarative rules is to start the correct VBA scripts.

The number of rules (IF statements) that appear in the scripts is approximately 650.

### 5.3 Organising the geometry

All geometry the KBES creates is stored in the CAD-model. Two different “geometrical sets” are used to organise the created geometry. A geometrical set is a group of geometrical features. All geometry that comprises the mid-surfaces for subsequent meshing is stored in a geometrical set called “thin.” Another set is called “thick”. Here, the sketches assumed to be used to form mid-surfaces in a later “pocket operation” are stored. This allows easy retrieval of the created geometry. A typical example of this is if a thin-walled tube is created. First, a solid shaft is drawn. The sketch does not contain any lines or arcs that are parallel and adjacent. The sketch is therefore stored in the geometry set thick. When the material inside the tube is removed in a later defined pocket feature, the KBES concludes that the sketch from the pocket and the sketch from the shaft together form a thin-walled structure. The medial lines and arcs are drawn, and an extruded surface with the same extrusion limits and sketch-plane as the pad is created in the geometry set thin. The sketch in thick is removed.

### 5.4 Surface idealisation

The object of the KBES is primarily to find the mid-surfaces of the thin-walled solids. In areas where the mid-surface is not meaningful to use, the solid geometry is kept or alternatively replaced with other types of surface representations.

The required idealisation depends largely on what meshing algorithm is to be used. It is possible to create mesh without making a surface first. One way is by extruding the mesh directly from the sketches. However, since several meshes often are included in the model, the trimming of these is facilitated by first making the surfaces.

Ideally, the created surfaces should be automatically meshable in the FE program. However, some manipulation to improve the element shapes is often needed. The “Advanced surface mesh” in CATIA V5 is used in this work. This means that the surfaces must be arranged in such a way that the meshing algorithm can identify and place nodes in the borders between the surface patches in a purposeful way.

## 5.5 Description of the function of the KBES

Below follows a brief description of how the KBES works. Please refer to Figure 2 which shows the basic components of the KBES:

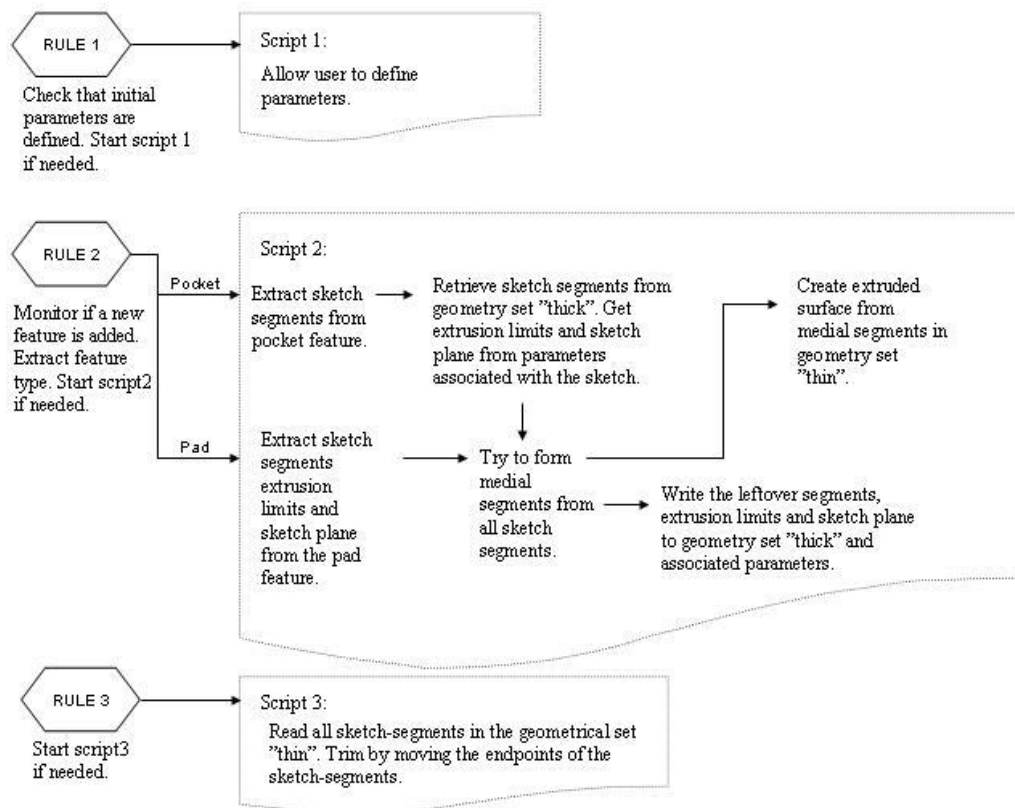


Figure 2: Main features of the KBES.

If the styling surface, the general material thickness, and the tuning parameters have not been defined, rule 1 is fired starting script 1 which allows the designer to define them. After the initial step, the system waits for the CAD-user to define the features of the solid geometry. When a feature is added, rule 2 is fired. Rule 2 checks what type of feature has been added. If the type is "Pad" or "Pocket", script 2 is run.

Script 2 collects the properties of the newly added feature. The feature type, the extrusion limits, the sketch plane, and the lines and arcs comprising the sketch on which the feature is based are included in the properties. The lines and arcs are here called the sketch-segments.

Should the added feature involve removal of material (i.e. a pocket), the sketch-segments in geometry set thick are retrieved and compared with the sketch-segments from the newly added pocket. If sketch-segments are found to run in parallel and be adjacent to each other, their mid-segments are calculated. The original feature's sketch plane and extrusion-limits are used to create an extruded surface in the geometry set thin.

If the added feature is a Pad, the mid-segments of the sketch (if any) are extracted and used to create a surface in the geometry set thin. The sketch-segments that could not find a parallel adjacent counterpart are stored in the geometry set thick.

Now untrimmed geometry exists in the geometry set thin. It causes rule 3 to fire. Rule 3 starts a script that reads all sketch-geometry in the geometry set. It identifies all intersections and gaps between all sketch-segments contained in the geometry set. It trims the geometry by moving the endpoints of the lines and arcs in the sketches. It also divides the sketch-segments if required.

Note that due to the parametric definition of the surfaces, they will be updated as a consequence of the change in the endpoints of their governing sketch-segments.

Figure 3 shows two typical situations where sketch-segments are trimmed. The sketch-segments have associated extruded surfaces. In situation 1, the intersection between the lines is calculated and the endpoints are moved across the gap to the intersection. Situation 2 requires that the line's endpoint be moved to the intersection and that the arc be divided into two arcs. Replacing the arc with a line would, of course, require that the line be divided. The division is made to ensure that the surface is split at the intersection. This facilitates the subsequent meshing. It is the parameter "general material thickness" that governs in what distances these trimmings are to be made. Tuning parameters, which can be adjusted by the user, providing the upper and lower borders of the range, are also used to tune in the trimming operations.

Note that the general material thickness is primarily used to provide a distance in which the different sketch-segments can be said to be adjacent. Since the mid-segments are calculated from the sketch-segments, the material thickness can be allowed to vary within a considerable range.

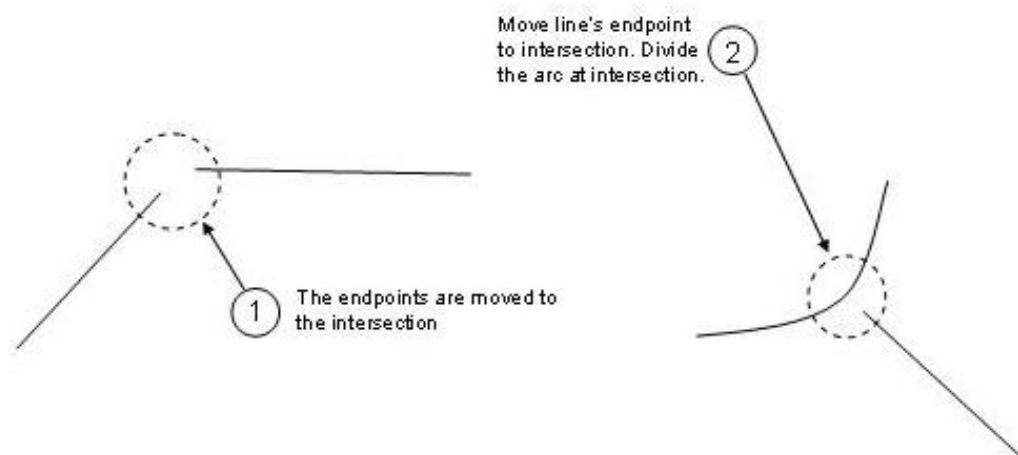


Figure 3: Trimming of adjacent segments.

## 5.6 Aid functions

The work described here has concentrated on the extraction of the mid-segments and their trimming. This part of the KBES can operate with full automation. However, the rules described will not provide a complete surface representation of a real life CAD-model. It would be possible to add rules that apply in situations where input is needed from the user. Those rules tell the user to manually perform, for instance, trimming operations with support from the KBES. This has less interest from a scientific point of view. Rather this work has concentrated on tasks that need extensive algorithmic procedures and can operate with full automation.

## 6. APPLICATION EXAMPLE

In Figure 4, a clutch housing for a garden-trimmer is shown. It is to be manufactured in aluminium by means of die-casting. This example will show how the solid model is built and the corresponding surface idealisation is created simultaneously.

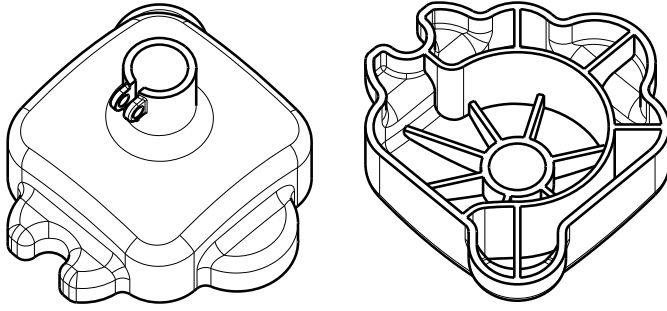


Figure 4: Clutch housing.

The user starts by drawing a pad according to Figure 5. A copy of the sketch is stored in the geometry set thick, and the associated parameters are set.

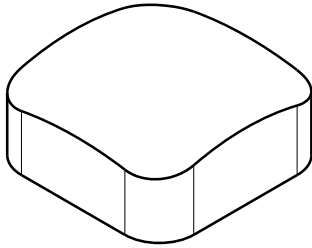


Figure 5: First feature "Pad".

A pocket is then created to remove the material inside the Pad. The first sketch is deleted from the geometry set thick. The two sketches are evaluated, and their mid-segments are created according to Figure 6.

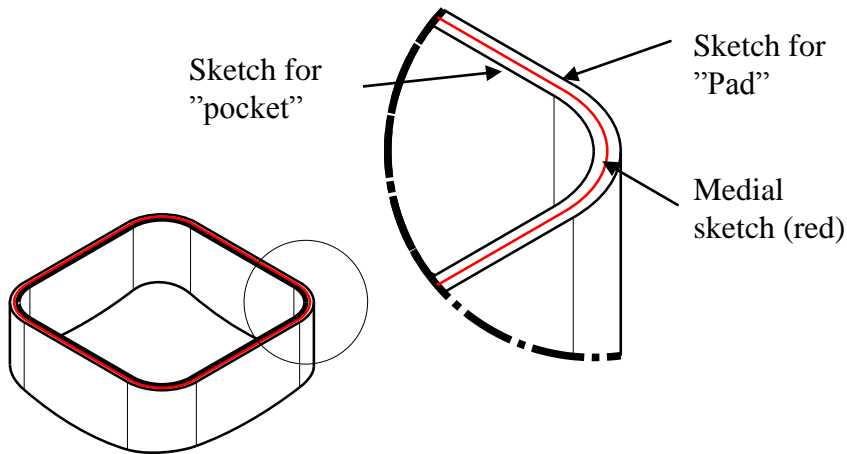


Figure 6: Second feature "pocket".

The mid-segments are used to create an extruded surface. The first feature's (the pad's) sketch-plane and extrusion limits are used to determine the extrusion limits of the extruded surface. In this case, the first limit is the offset styling surface, and the second limit is zero.

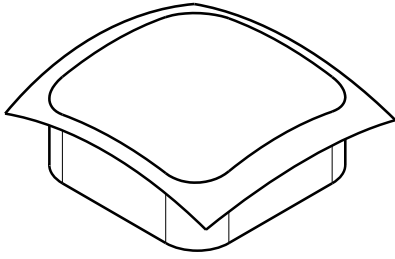


Figure 7: Two intersecting surfaces present in the geometry idealisation.

The styling surface and the created surface need to be trimmed. A fillet is also added on the edge of the resulting surface, as seen in Figure 8. This is done manually.

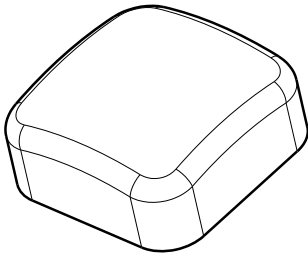


Figure 8: Resulting surface after trimming.

Next, the three protrusions on the sides of the Pad are created, as seen in Figure 9.

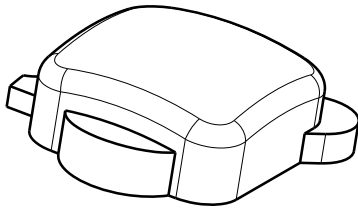


Figure 9: Result after adding three protrusions.

After the system has identified the medial segments, it concludes that they must be trimmed according to the principals described in Section 5.5. The trimming operation for one of the protrusions is illustrated in Figure 10. Note that manual trimming of the surface patch is also needed.

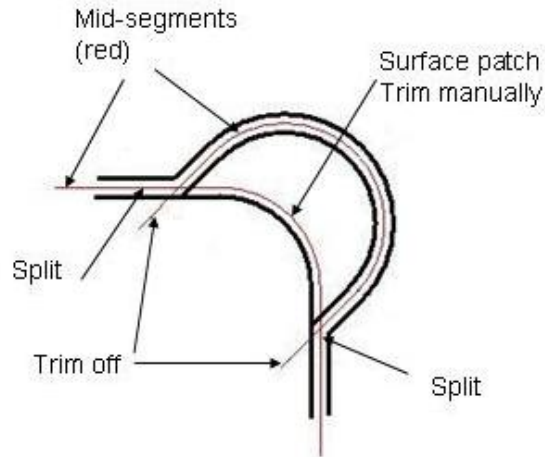


Figure 10: Trimming of segments.

The user then draws the attachment for the shaft. Rule 2 is fired, and the medial surface is created. The hole in the styling surface needs to be created manually.

Next, the inner structure of stiffeners is sketched. The sketches are directly identified as thin-walled, and an extruded surface is created from the mid-segments. They are also trimmed against the existing extruded surfaces.

The user then makes the flanges on the attachment. The flanges are identified as thin-walled. The mid-segments are created, and surfaces are extruded from them and trimmed against the existing thin geometry.

The final result of the operations is shown in Figure 11. Figure 11 also shows the geometry meshed using 2mm shell elements.

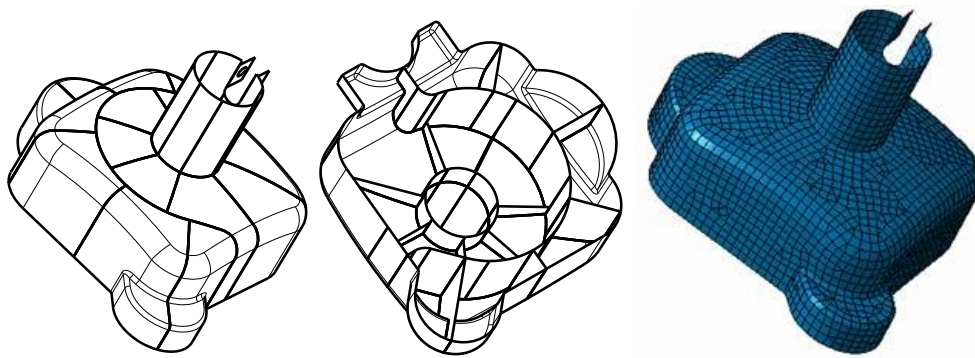


Figure 11: Final surface idealization.

Meshed using 2mm shell elements

## 7. CONCLUSIONS AND FURTHER WORK

An early prototype of a CAD-integrated KBES for the concurrent creation of a shell idealisation of thin-walled solid model has been presented. The KBES operates by interpreting the properties of the features added to the CAD-model. It creates a geometrical idealisation in the same CAD-model from this information.

The main advantage of the presented method, compared with using KBES containing higher-level engineering objects or general algorithms for creating surface idealisations, is that no prior definition of these engineering objects has to be made (applicable in the case of KBES). The

idealisation can also be accessed at all times in different states of completeness for use in FEA. It is easy to change the geometry idealization due to its parameterised definition. Furthermore, the designer can check that the created geometry is correct and change it if necessary.

It is the intention of the authors to further develop this method and evaluate its usefulness in practical applications. This has to be governed by the modelling technique used in the studied application. The impact on the time and resources spent on the preparation of models for FEA must be further evaluated.

The KBES described has concentrated on the extraction of the mid-surfaces. The processing of features in which the mid-surface is not defined or not suitable is also to be addressed.

Finally, since the number of IF-statements is large even in this early prototype, the possibility of using an increased number of declarative rules must be explored. This will probably decrease the number of IF-statements the KBES needs to evaluate thus, enhancing both manageability and the performance of the code.

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# A CAD-INTEGRATED SYSTEM FOR THE AUTOMATED IDEALIZATION OF CAD- MODELS FOR FINITE ELEMENT ANALYSIS

ROLAND STOLT

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## ABSTRACT

The work described in this paper seeks to minimize the time spent on manually reducing thin-walled CAD-geometry into surface idealizations. The purpose of the geometrical idealizations is the creation of shell element meshes for FE-calculations. This is motivated by time and thereby cost savings. It is also motivated by the desire to make the results of the calculations available earlier in the product development process, thus allowing the results to guide the designs to a larger extent.

Systems for automated geometry idealization and the creation of FE-models already exist. However, this paper describes a novel approach with the working principle of analyzing how the CAD-specific features of the CAD-file history tree are constituted. This information is used to automatically create the best practice geometrical idealization in the same CAD-model. An evaluation of the performance of the system in an industrial example is also presented.

Keywords: KBE, CAD, Finite element analysis, Simulation-Driven design, and CATIA V5

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## 1. INTRODUCTION

Today simulations are often used to verify proposed designs. However, in order to provide more support for the product development process, the simulations need to be more closely integrated into all stages of the product development process. This is commonly referred to as simulation-driven design (Sellgren 1999).

To meet these expectations closer, integration between the CAD and CAE tools is needed. An attempt to formulate what components such system will need is presented in (Shephard, Beall et al. 2004). Apart from keeping track of the simulations performed, the systems need to assist in the creation of simulation models, possibly through the automated processing of CAD-models.

It is the task of deriving the simulation model and, in particular, the FEA-model (Finite Element Analysis) that this work seeks to address. Starting with the CAD-model, the process involves the idealization of the geometry to fit the purpose of building the FEA-model.

The geometrical idealizations fall into two major categories (Dabke, Prabhakar et al. 1994):

- The suppression of details that have a limited impact on the simulation results.

- The dimensional reduction of the solid geometry into lower dimensional entities.

An example of the former is the suppression of fillets or holes that will only have a local effect, and, consequently, have a low impact on the global results. The latter is exemplified by the reduction of thin-walled structures into shell elements assuming that the conditions through the thin wall remain constant. Another example is when part of the geometry is replaced by a lumped mass. In all cases, the overall objective is to reduce the time required to solve the problem by concentrating on the essential aspects of the analysis.

The time spent on idealizing CAD-models for the purpose of creating FE-models accounts for a large part of the manual work put into the FEA. Since simulation is expected to play an important role in future product development and the time available in the product development process decreases as customer demands for new products increase, much can be gained from automating the process where possible or at least providing help for the analysts.

It can be argued that in recent years the FE-programs and computing power have increased such that the number of FE-problems that can be solved using the CAD-model directly without any prior idealization has increased. However, the development has also led to increased demands for more advanced problems to be solved. This will make it a necessity to idealize the geometry in the advanced problems to keep the time required to solve the problems at a manageable level. It may also be desirable to solve an FEA problem a number of times under the variation of the design parameters exploring the entire design space, so that an optimized product can be obtained. In this case, the computing time would, in the advanced cases, simply be too long if the CAD-geometry had not been subjected to prior idealization.

Furthermore, the object of the FEA may vary throughout the product development process so that several idealizations with a varying degree of fidelity need to be created. This places an increased demand on the rapidity of the process of creating the idealizations.

There has consequently been much effort put into automating the creation of the FE-models. One of the more notable examples of research in this matter is conducted at QUB (Queens University of Belfast) (QUB 2004). The idea is to use the shape description of the CAD-model provided by the MAT (Medial Axis Transform) to identify regions where dimensional reduction or feature suppression can be made. After identification, FE-mesh with appropriate, and possibly mixed, dimensions is automatically created (Armstrong, Donaghy et al. 1996), (Armstrong, Bridgett et al. 1998). (Lee, Price et al. 2003) show how the MAT can be used to identify regions of unwanted topology in CAD-models. The topology is corrected using topological operators leaving the CAD-models ready for FE-meshing.

In (Joshi N. 2002), algorithms for automatically detecting and suppressing hole and fillet features in surface models are presented. The objective is to simplify the models so that the subsequent meshing is facilitated.

In order to make an appropriate choice of element type and a reasonable geometry simplification, the purpose of the geometry must be known (i.e. the design intent) (Ansaldi 2003). The way in which the load is intended to be applied to the structure is an example of such design intent.

KBES (Knowledge Based Engineering Systems) is the mainstream way of keeping the knowledge concerning how to make the most purposeful FE-model stored and applicable in various design situations. In a KBES, the knowledge is represented in the form of design rules and algorithmic procedures where applicable. This will narrow the usability of the system to be specific to a certain category of products sharing the same design intent. Furthermore, maintenance of the system will be necessary, as the design rules and algorithmic procedures have to be held updated as the corporate knowledge increases. New designs may also have an impact on the design rules.

There are a lot of examples in the literature where KBES has been successfully used to keep FE-models available and automatically adapt them to new variants of former designs (Chapman and Pinfold 2001; Isaksson 2003; La Rocca, Krakkers et al. 2003).

The automated adaptation of previously defined FE-models is limited to relatively small variations in the designs. Instead, the work described in this paper seeks to instead build a system that creates the FE-model from scratch, under the restrictions that the design intent and production method remain unchanged.

One of the major characteristics of a KBES is that the rule-base can be modified or extended without having to change the code in the controller (in other words, the rule-base is separated from the inference-engine). Although the system described in this paper has a small rule-base, defined in CATIA V5 knowledgware, the character of the program is predominantly procedural.

The purpose of the few rules is primarily to check that a number of initial conditions are fulfilled and to trigger the evaluation of the features of the CAD-model and the subsequent adding of surface representations. The rule-base is merely a convenient way of structuring the coding. To add, revise, or delete rules will require knowledge about the scripts, making the system unsuitable for user maintenance.

The CAD-system allows easy access to the properties of the features in the construction history tree of the CAD-file. These properties include, for example, the appearance of the sketches i.e. the coordinates of the arcs and lines which the features are based on. Information is also extracted from the extrusion limits and the sketch planes of the features.

It has not yet been possible to extract sufficient information to allow the automated creation of the geometrical idealization in all design situations encountered in the studied application. In these cases, the system has to allow help to be provided by the analyst. Where possible, automated procedures are used. Otherwise, the analyst is asked to support the system with the necessary information. The paper focuses on the reduction of the solid CAD-geometry into surface representations for subsequent meshing. Detail suppression will be a consequence of the described method.

The described way of defining the geometrical idealization will complement the two mainstream ways of automatically creating geometrical idealizations and FE-models. The proposed system displays a lower degree of automation. Further, it does not grasp the whole process of creating the FE-model. However, it is thought to provide more flexibility in that the predefined objects are defined at a lower level than in the referenced KBES approaches.

The intention is to allow the term “product variant” to be somewhat expanded as described in the following sections.

## 2. CASE STUDIED

The aim of this study is to develop a prototype system with the working principle described above and to evaluate its usefulness in real-life design situations.

A local manufacturer of forest and gardening equipment provides these real-life design situations. The products are mainly chain-saws, trimmers, and lawnmowers. In order to secure that the final products are not damaged by the vibratory excitations from the engines, FEA is conducted on the proposed designs. The products consist of a large number of parts manufactured mainly through die-casting and injection molding. These production methods imply that the parts are thin-walled and have defined tooling draft directions. Both are prerequisites for the described system. Prior to building a complete FE-model of the products, each part is idealized mainly by reducing the thin-walled geometry into a surface-model that is later meshed using shell elements. After

idealizing all parts, they are assembled into an FE-model of the complete product and subjected to a recorded frequency spectrum of engine vibration. The risk of structural damage is estimated from this model.

The process of transferring the solid geometry to its shell-element representation is currently done manually in the CAD-system by drawing wire-frame and surfaces “on top” of the existing solid geometry for each part. These surfaces are later meshed using shell elements.

There have been efforts made to use the pre-processing capabilities of commercial FE and CAE systems to extract the mid-surfaces, but the parts contain areas where the mid-surface is not defined or unsuitable. As such, the most straightforward method seems to be the manual creation of the surfaces.

An appropriate remark is that there are other techniques of mesh creation available in the commercial FE and CAE-systems that would eliminate the need for first making a target surface for the mesh. A possibility is to use vertices, faces and edges from the original geometry to govern the mesh creation. In the application studied, prior surface creation has been found to be the most efficient way.

The procedure of manually creating the surface for meshing is considered highly time consuming. The time spent may be as much as three man-weeks for the most advanced parts. Consequently, the results of the FE-calculations are available late in the product development process. This limits the possibilities of making changes to the design.

If the time spent on preparation for the FE-calculations could be reduced, more time could be spent on more value-adding activities, such as optimization. The adoption of the concept of simulation-driven design would also be largely facilitated. The ultimate goal of the system is to assist in the creation of the complete FE-model. However, to start with the creation of the surface representation ready for meshing has been addressed because it has been identified as the most laborious and time-consuming task of the FEA.

It has not, as earlier mentioned, been possible to create a system capable of creating FE-models with full automation in all encountered design situations. Furthermore, if tried, the cost of developing such a system would be high since a very large number of possible situations have to be taken into account. Since the system’s capability is limited to a specific type of products, the effort would probably be hard to motivate. It is therefore the task of this research to find a reasonable level of automation specific to the application. As a consequence, the system will not be able to eliminate the need for having an expert in the studied FE-calculations check the created geometry and make corrections if needed. However, the corrections are facilitated due to the fact that the system will provide a parameterized surface idealization that is more easily changed than a surface idealization or mesh provided in a neutral format.

### 3. THE PROPOSED SYSTEM

The first version of the system described in (Paper I) works by registering the features as they are added by the designer. When a feature of the type “Pad” or “Pocket” (methods in CATIA V5 for adding and removing material by means of linear extrusions) is added, the system collects the added feature’s sketch, sketch-plane, and extrusion-limits.

The lines and arcs comprising the sketches are checked to see if any of them run in parallel roughly one material thickness apart. If such entities are found, their mid-segments are calculated. These segments are used to create an extruded surface with the same extrusion-limits and sketch-plane as the original feature. The entities in the sketch that could not be paired with a counterpart are stored. It is assumed that a later defined pocket operation will make the structure thin-walled. When a pocket feature is added, the sketch of the pocket is compared with the

stored sketch-segments from earlier runs. If the sketch-segments can be paired, their mid-segments are calculated and used to create surfaces with the same sketch plane and extrusion limits as the original feature.

The created geometry is a “2.5D” mid-surface representation oriented in the tooling draft direction. It is assumed that the CAD-file from start contains a “styling surface” providing the outer appearance of the part. It is this surface, offset by half the material thickness, that provides the extrusion limit in one direction of the 2.5D rib structure.

To illustrate how the program steps through a CAD-model, a simple part with four features as shown in Figure 1d is to be processed.

First, a pad is extruded from sketch 1 as seen in Figure 1a. No thin-walled entities are found so the sketch is stored until the pocket feature based on sketch 2 is defined (Figure 1b). The mid-segments from sketches 1 and 2 are calculated, and an extruded surface with the same extrusion limits as the first feature is created. Next, the crossing inner rib structure is extruded from sketch 3 of Figure 1c. The mid-segments of sketch 3 are calculated and used to create an extruded surface. Finally, the mid-segment of the two concentric circles of sketch 4, comprising the sketch of the circular rib of Figure 1d, is calculated. The resulting extruded mid-surfaces from the different steps are seen in Figure 2 b-d. Note that, as new surfaces are being added, the mid-segments are split if they intersect with previously added segments. Likewise, the endpoints are moved to the intersection point if a gap in the magnitude of one material thickness should occur. This is done to make sure that the resulting surfaces are connected and split at their intersections to facilitate the subsequent meshing.

Due to the parametric definition of the surface idealization, it will be updated (at every model update) to follow the revised location of the endpoints as well as the additional points added in the splitting operations.

To finalize the surface representation, the offset styling surface has to be manually trimmed against the created surfaces as seen in Figure 2d. A surface representation ready for meshing has now been automatically created, with the exception of the manual trimming.

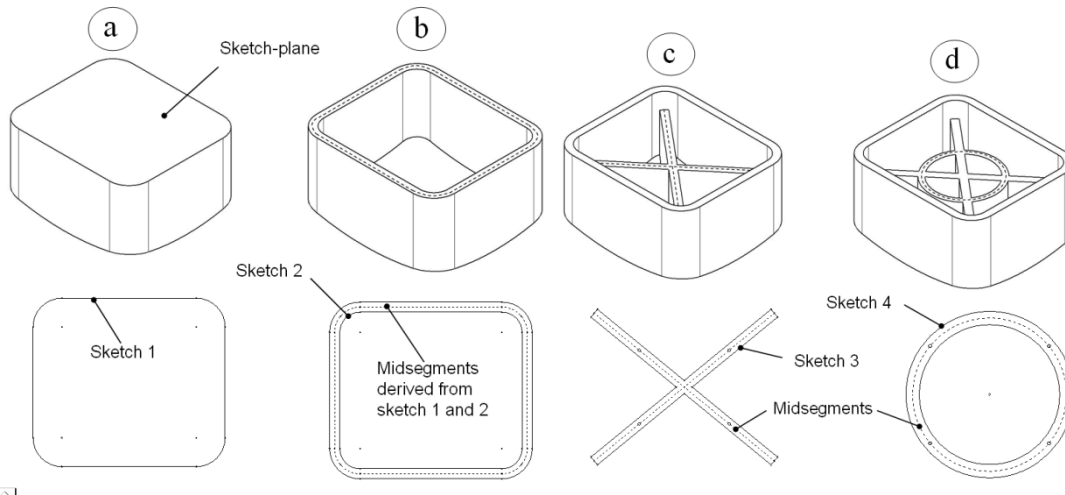


Figure 1: Steps a to d.



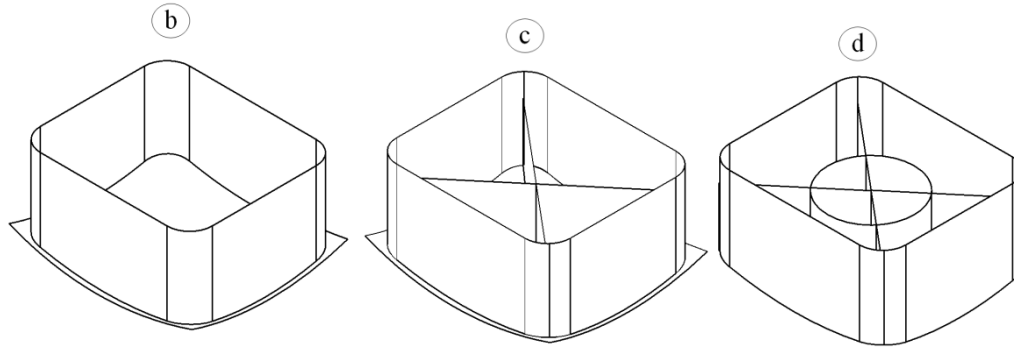


Figure 2: Resulting mid-surface representation.

The described way of creating a surface ready for FE-meshing has been found to have three major advantages:

- The created geometry is parametric, and can be associated with the original geometry of the created geometry, which makes it easily changed.
- The designer/FEA-expert can check and correct the created geometry as it is created.
- Having the surface representation in the same CAD-model is thought to facilitate storage.

#### 4. EVALUATION OF THE SYSTEM

It was discovered during the evaluation of the system that the method of processing each feature as the designer defines it, which was the original intent, is perceived as disturbing to the designer's work. It was found that it is better to "post-process" the model. This means that the system is assigned to work on the completed CATIA-file starting with the first feature in the history tree and process the features one by one. An expert in the FE-calculations can be assigned to monitor this process and provide the necessary information and make corrections to the created geometry.

A simple test has been performed to indicate the time saved by using the system. The simplified CAD-model of a clutch housing intended for die-casting as seen in Figure 3 was selected as a test object.

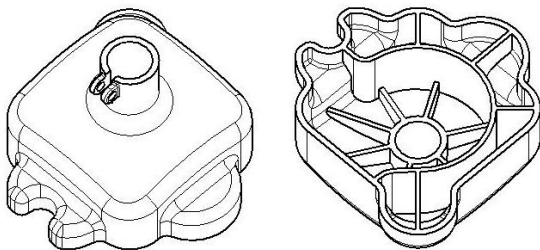


Figure 3: Test object to estimate time consumption from solid-model to FE-mesh.

The surface representation of the part was first manually created in the same manner as described in Section 2. The time spent from start to complete mesh was 73 minutes and 27 seconds. With the system switched on, the time spent to create the wire-frame is near zero. The system required 8 seconds to draw the extruded surfaces. The time saved in this example is 38 minutes (52% of the total time). See Table 1 for details.

Task	Manual m:s	System m:s
Create wire-frame	28:53	~0
Extrude surfaces	8:15	0:8
Trim surfaces	26:27	26:27
Create mesh	9:52	9:52
Total	73:27	35:27

Table 1: Time spent to complete FE-model.

As seen, it is the creation of the wire-frame that accounted for most of the time spent and, consequently, yielded the largest time saved.

Note that the test object was simplified. Therefore, much less time was spent than in a real-life situation.

## 5. FURTHER DEVELOPMENT OF THE SYSTEM

The first version of the system was aimed at identifying thin-walled structures in the sketches and creating the mid-segments and surfaces. This is not sufficient, since real-life modeling also involves the removal of material. Furthermore, in some of the areas of the studied CAD-models, it is not possible to derive any mid-surfaces or they may be unsuitable for building the FE-model. Here, surface representations based on corporate FEA experience are stored in the system and inserted where applicable.

The removal of material from the solid-model is done in two ways. In the first is the way (described in Section 3), a pocket is added to remove material, thereby creating together with an earlier defined pad a thin-walled section (in other words, material is removed from the thick parts of the model). In the second way, material is removed from the thin-walled geometry, for example, when creating a hole or removing part of a rib. In these cases, a corresponding portion of the extruded surfaces also needs to be removed. The way the system handles these features is to capture their sketches and extrude “help-surfaces” from them. They are called help-surfaces here, since they are only used to remove the part of the surfaces they enclose. Unlike the extruded surfaces replacing the thin-walled sections that are always oriented in the tooling draft direction, the help-surfaces may have a direction, perpendicular or parallel to the tooling draft direction. The extrusion limits of the solid body features assigned to remove material often consist of faces in the solid model (up to surface) or other features (up to next).

Provided the axis of the pocket or hole-feature is oriented in the tooling draft and one extrusion limit is located beyond (through all) the styling surface, the corresponding patch is consequently removed from it.

Should the orientation of the axis be perpendicular to the tooling draft direction, it has turned out to be difficult to handle non-dimensional extrusion limits. It is currently handled by the system through seeking in roughly one material distance from the sketch plane. Examining the coordinates of the sketches of the extruded surfaces does this. If a surface is found, the patch enclosed by the help-surface will be removed. This limits the permissible modelling-technique to keeping the sketch plane near to the target geometry. Alternative ways of handling the non-dimensional extrusion-limits may exist. However, they have not yet been explored.

To handle the areas where it is not possible to derive any mid-surface, the solution explored is to define “standard elements” closely related to the function of the part or the production method. Examples of such standard elements are the attachments for the screws connecting the chainsaw sword to the crankshaft housing and the “towers” used to make contact with the ejector-pins of the tooling assembly. The ejector-pins are used to eject the finished part from the tool. They exert a force on the part, which, consequently, will be slightly deformed.

The purpose of the towers is to secure that the damage is placed on them and not in an area relevant to the function of the part. Also, the interface for attaching the cylinder head can be regarded as a standard element. These standard elements typically do not have any sketch segments that can directly be regarded as thin-walled. However, since they are assumed to be modeled in a similar way in all corporate CAD-models, they can be identified by the system through their standardized dimensions of the feature sketches. It is established company practice how these features are best idealized. The predefined idealizations are stored within the system and retrieved and inserted in the complete idealization. The orientation of the inserted idealization is determined from the neighboring sketch-segments of the already added surface representations.

This is a deviation from the idea of not having any predefined representations stored. However, the usability of the system increases if it allows such standard elements to be defined when needed in the application. This allows a more complete surface representation to be obtained. Furthermore, the complexity and completeness of these representations are far less than those in the KBES approaches of the referenced work.

To exemplify the removal of surface patches resulting from pocket and hole operations and the insertion of standard features, one side of a crank-shaft housing is chosen as an example, as seen in Figure 4. The part is manufactured by die-casting in magnesium and is currently in production.

First, the system identifies the standard diameter of the four ejector-pin towers using the sketches on which they are based. One of them is shown in detail B of Figure 4. The system inserts a surface representation that, meshed with shell elements, has mass and stiffness that approximates the original ejector-pin towers. The standard dimensions of the hole through the rectangular box identify the screw attachments. The solid geometry is kept for this standard element (Detail A). This leads to the final FE-model being a hybrid, containing both solid and shell elements. A hole is created in the tooling draft direction as seen in Detail C, since the extrusion limit is “through all” a circular patch is removed from the styling surface. Detail D shows that part of the surface representing the thin-walled rib is removed using a help-surface. Note that the surface representing the rib has been created by the system earlier. Therefore, which part of the surface representation to remove is determined by finding which sketch-segment is located within the range of one material thickness from the features sketch-plane and can be pierced by the help-surface.

In order to define new standard dimensions for the predefined elements, the system allows the designer to name them in the history tree. If, for example, a new type of attachment-screw is to be used, a hole with the desired diameter should be added. If the feature is named in a company-specific way in the history tree, the set of standard diameters stored in the system will consequently be updated. This will allow an idealization with the new dimension to be inserted into the model.

## 6. DETAIL SUPPRESSION

As mentioned in the introduction, the proposed system is designed to reduce the solid geometry into surfaces for meshing. In the CAD-models, there may be features that affect the results of the

analysis very little, yet causing the computing time required to solve the problem to increase significantly. These features should not be represented in the surface idealization, provided the error that this introduces is managed. Since features like fillets are not recognized by the system, they will not appear in the created surface idealization, in a sense providing feature suppression. The system allows a threshold value for the maximum fillet radius that can be ignored to be entered. If the value is exceeded, the analyst is asked to add the fillet in the surface representation manually. It should be noted that similar functions are provided in commercial FE and CAE systems allowing, for example, the analyst to define the minimum hole diameter to be taken into account by the meshing algorithm.

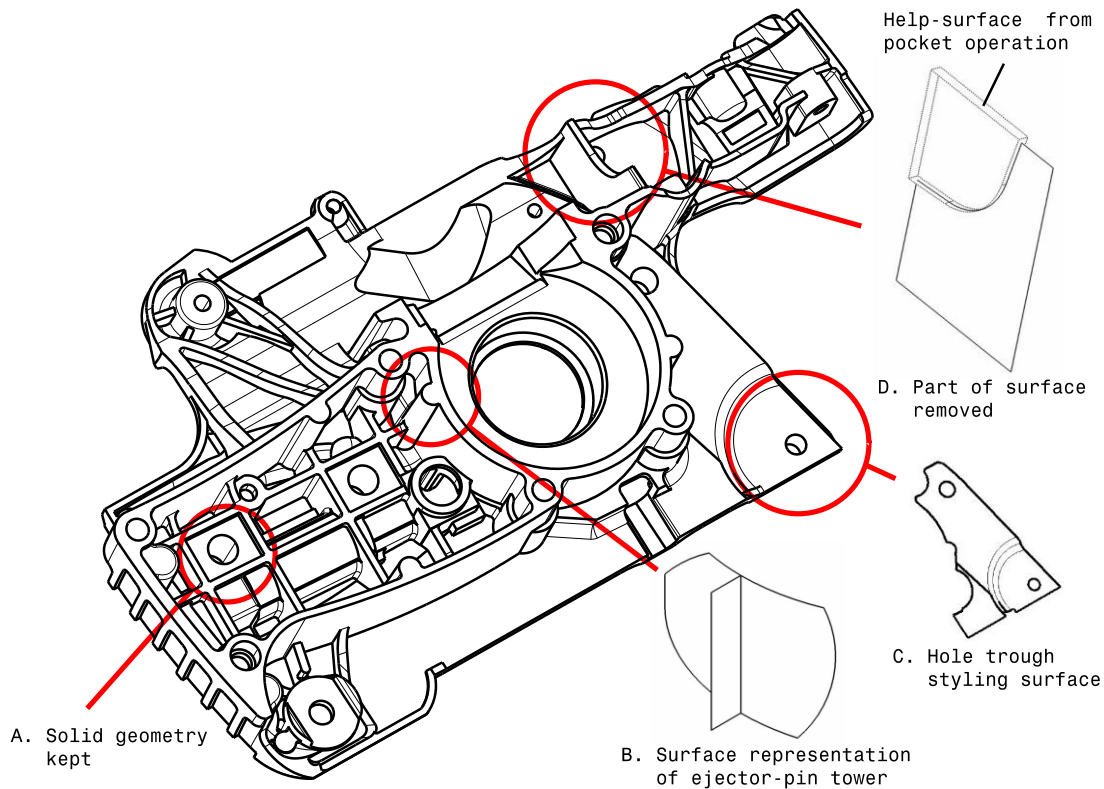


Figure 4: Standard elements and surface patch removal.

## 7. PROGRAMMING

There are a few rules defined in CATIA V5 knowledgeware that are evaluated at every knowledge-base update. Their primary purpose is to read the construction history tree of the CAD-model and to start the correct procedures written in VBA (Visual Basic for Applications) accessed directly through the CAD-system.

Since the CAD-system allows the easy retrieval of the features of the CAD-models through scripting, no feature recognition technique directly from the CAD-file is necessary.

A programming example is given below to illustrate how the features of the CAD-model are accessed, how the correct scripts are started, and how the derived surface representation is inserted in the model.

Rule1, which resides in the CAD-system, selects the features (shapes) belonging to the solid geometry of the CAD-model (PartBody). Depending on the type of feature, different scripts are started. Should the feature, for example be a “Pad” it is passed on to script 1, as seen below, where the sketch-plane, the extrusion limits and the geometrical items of the sketch are extracted.

```

*/Rule 1:
Set Body1=PartBody
*/PartBody contains the solid geometry
of the CAD-model.
Set Shapes1=Body1.Shapes
*/Shapes1 is the collection of features in
the PartBody
For i=1 to Shapes1.count
*/count is the number of features in the
collection.
Shape1=Shapes1.Item(i)
*/The feature # i is extracted
If Shape1.Type="Pad" then
script1(Shape1)
If Shape1.Type="Pocket" then
script2(Shape1)
...      ...
      ....
If Shape1.Type="Fillet" then
scriptn(Shape1)
*/ The appropriate script is started,
depending the feature type
Next
End rule 1

```

```

Script1(Shape1)
Set Plane1=Shape1.SketchPlane
*/The SketchPlane of the feature is
extracted
Set Limits= Shape1.Limits
*/ The extrusion limits are extracted
Set sketch1=Shape1.Sketch1
Set
GeomItems1=sketch1.GeometricalItems
*/The geometrical items in the sketch are
extracted
StdFeat=ExtractStdFeat(GeomItems1,
Limits)
*/ Check if it is a standard feature.
If StdFeat=FALSE then
Wireframe=ExtractWire(GeomItems1)
*/ the wire-frame geometry is derived,
provided it is not a standard feature
Call WriteSurface(Wireframe, Limits,
Plane1)
*/ An extruded surface is created in the
same CAD-model
End Script1

```

Script 1 accesses the feature properties. Provided the feature is not one of the standard features described in Section 5, the procedure “ExtractWire” derives a wire-frame representation using the geometrical items (i.e. the coordinates of the points of the lines and arcs of the feature’s sketch). The procedure basically seeks in directions perpendicular to every sketch segment to see if there are any segments that run in parallel approximately one material thickness apart. If such segments are found, the mid-segment location is calculated.

Next, the derived mid-segments, extrusion limits and sketch-plane are used to create a surface representation in the same CAD-model, employing the procedure called “WriteSurface”.

Every time a new surface is added to the set of surface representations, they are trimmed against each other using the method described in Section 3. This trimming is governed by a script and triggered by a rule in the rule-base.

The nature of the algorithms of the program is to test all possibilities that resulted in a large number of IF-statements in the code.

The program structure can be viewed in the UML (Unified Modelling Language) (Booch, Rumbaugh et al. 1999) diagram of Figure 5, where the key features of the class diagram are shown. These classes are not to be confused with those of the CAD-system. The figure shows only the classes that exist in the scripts. The methods of the class “feature” are, for example, the same as described in script 1.

All communication with the CAD-system is handled through the interfaces shown in the figure.

The structure of the program is object oriented. The method of presenting the problem using UML schemes turned out to be helpful when designing and writing the code.

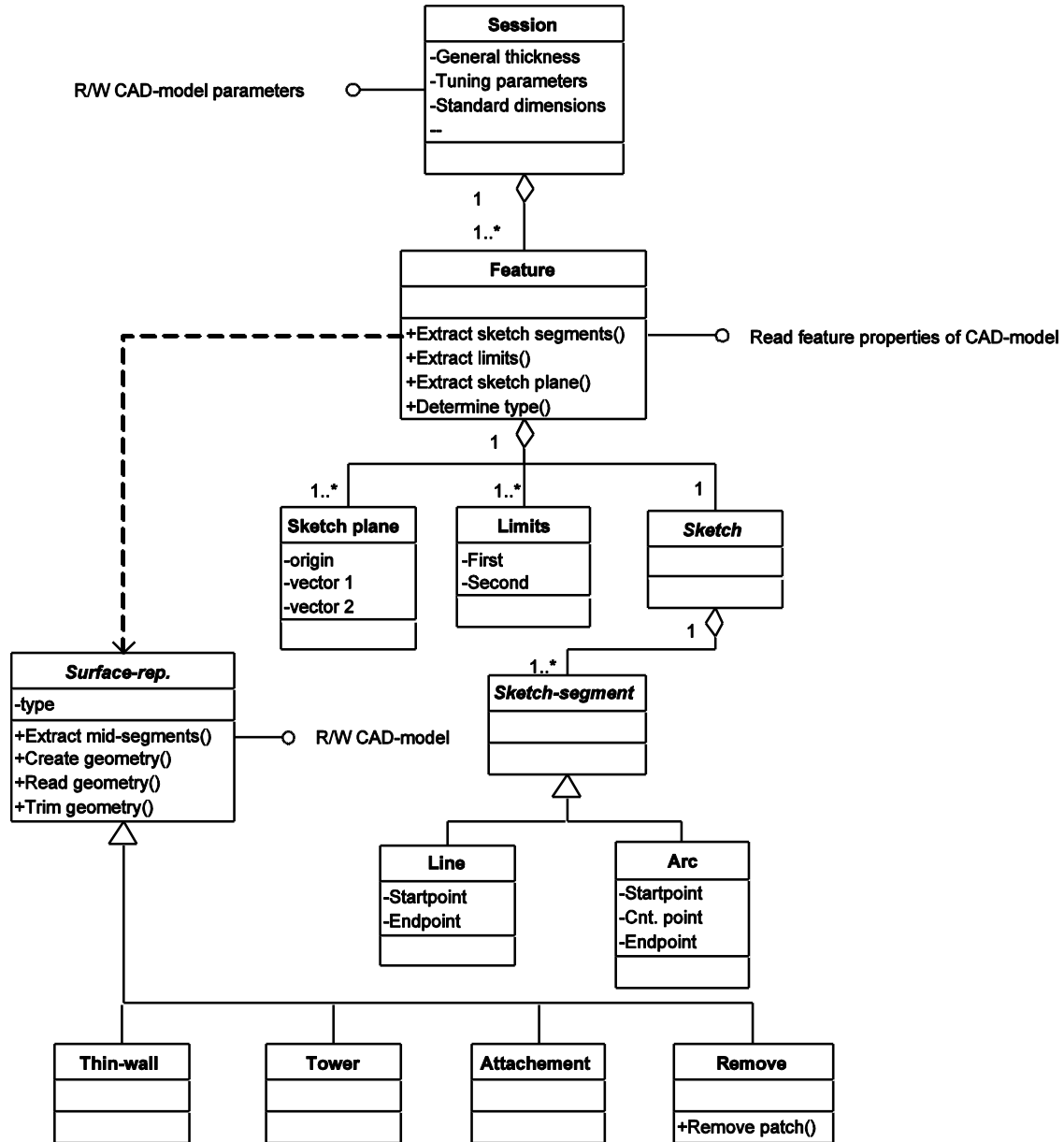


Figure 5: Simplified UML class diagram.

## 8. IDEALIZATION ERROR IN THE FE-MODEL

The idealizations made produce an error that, of course, must be handled. The criterion used for the creation of the surface idealization is the company practice that if the surfaces are completely covered by the original thin-walled solid geometry, they can be used to create an FE-model of sufficient accuracy.

Naturally, the task of the system should not be limited to the creation of the surfaces, but also include the mesh creation. In the real-life parts, the surfaces are drafted as needed to allow ejection from the tooling. This means that the material thickness is varying. The walls are broader at the base than in the top. Consequently, the shell thickness needs to vary. This is currently handled in the process of assigning thickness to the shell elements, a time-consuming part of the work not accounted for in the test presented in Section 4. Since the drafts have significant impact both on the mass distribution and the stiffness of the structure, this has to be

addressed by a system for processing real-life parts. The drafts are defined as features in the construction history tree, so the information on draft angle and affected feature is available for the system to acquire.

## 9. GENERALITY OF APPROACH

The described system is restricted to work on CATIA V5 files that have identifiable features in the construction history tree. Consequently, imported models will not be processable.

If there is a sudden change of CAD-systems within the company, much effort will have to be put into transferring the knowledge into the new system. However, there is ongoing research to make the translation of the parameterization between different CAD-systems possible (Ansaldi 2003) and (Mun, Han et al. 2003). The idea presented by (Mun, Han et al. 2003) recognizes the fact that modeling techniques for the creation of B-rep. models do not vary much from the commonly used CAD-systems available on the market. Therefore, the effort of creating a translation system is thought to be rather limited. This holds a promise that systems for translating the construction history tree between different CAD-systems will be available in the near future. This will allow an increased number of CAD-models to be processed using the proposed method. Alternatively, the system can be redesigned to handle the neutral file directly, provided the sketches are included in the neutral file.

## 10. CONCLUSIONS AND FUTURE WORK

It is possible to create a surface representation apt for shell-element meshing using algorithmic procedures and rules defined in a commercial CAD-system. Extracting the properties of the features in the construction history tree and using them to create a surface representation in the same CAD-model accomplishes this.

This approach is to the author's best knowledge novel. The method provides a complement to the well-known KBES and fully automated algorithmic approaches for creating a surface representation ready for the meshing of thin-walled solid-models.

It was found that the quite a bit could be gained from automating the creation of the wire-frame and the extrusion of the surfaces, both in terms of saving time and the effort required to define the system. The whole process of creating the FE-model was not grasped. If tried, it is thought that, since the system is specialized to a certain type of geometry, the cost of developing such a system will be too high. It has therefore been identified as a future task to find a reasonable level of automation given the effort required to develop the system.

Currently, the system is restricted to handling native CATIA V5 files only. Furthermore, the system does not aid in the assigning of thickness to the shell elements, which represents a large part of the total time spent on FEA.

Ideas on how to address the portability of the knowledge gathered and how to take the effect of the draft features of the CAD-model into account in the FE-model have been presented. It is the intention of the author to develop the ideas and examine the feasibility of incorporating them into the prototype system.

## 11. ACKNOWLEDGMENTS

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# REUSING CAD-MODELS OF DIE-CAST PRODUCTS FOR FEA

ROLAND STOLT

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## ABSTRACT

Parts for trimmers, lawnmowers, and chainsaws are often manufactured by die-casting in light alloys. These parts just as often have a complex geometry, with thin-walled regions mixed with regions of non thin-walled geometry. This paper discusses the process of creating FEA-models from CAD-models of such parts. The objective of the FEA in the particular case described in this paper is to predict the performance of the complete products. Since a FEA-model of the complete product requires extensive computational power to solve, it is necessary to keep the computational cost down. One way is by using shell-elements instead of solid-elements where applicable.

The creation of the shell-element mesh based on the solid CAD-model can be time-consuming, especially if the CAD-model is complicated, as is often the case in real products. To aid in the creation of the shell-element mesh, commercial software for the extraction of the mid-surfaces in the CAD-model is available. In this paper, two different software programs for mid-surface extraction are tested, both included in pre-processors for high-end FEA solvers. The software was tested on six different die-cast or injection moulded parts, all components in consumer products available on the market.

The result is that the software can extract the individual mid-surfaces well, but the connections between them are often not created properly.

Furthermore, the software cannot, as expected, deal with the regions that divert too much from being thin-walled. This paper presents a two-part idea, regarding how to treat these regions. First, suitable representations from a depository of representations known from experience to represent the geometry well are retrieved. Second, they are inserted in the appropriate position in the FEA-model automatically.

A demonstrator program created in CATIA V5 using VBA is also presented in this paper to show the feasibility of the idea.

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## 1. INTRODUCTION

The tasks of reusing solid CAD-models with thin-walled geometry for meshing with shell elements have been around for a long time. Although the geometry already exists, it is necessary, depending on the type of analysis to be performed, to do various types of time-consuming pre-processing. This pre-processing may include removing features of low relevance and creating target surfaces for the shell element mesh. Since the geometry already exists, it should be possible

to automate or at least provide help for the analyst in the pre-processing step. The vendors of CAE software have acknowledged this need, and provide tools for this type of pre-processing. One commonly used method of dealing with such problems is to use a tool for mid-surface extraction. Such tools are often provided in high-end pre-processing programs for FEA. The created mid-surface can be used as a target surface when meshing with shell elements.

This paper seeks to evaluate the performance of such mid-surface extractors on geometries intended for production by die-casting or injection moulding. A manufacturer of foresting and gardening equipment (such as chainsaws and lawn mowers) where injection moulded and die-cast parts are commonly used has initiated the research presented in this paper.

In this type of geometry, the production method implies that the parts are fairly thin-walled in order to give a reasonable production cycle time. In addition, these types of parts are in most cases geometrically complex, featuring drafts, parting lines, etc. This is to make a well-working and reliable tooling.

One circumstance that further complicates the picture is that die-cast and injection-moulded parts often consist of mixed geometries. In other words, although they are thin-walled in most regions, there are other regions that do not lend themselves easily to approximation in shell-elements. In these regions, automated mid-surface extractions cannot be expected to work. They merely find the regions where the boundary walls in the model describe thin-walled sections. In order to deal with these regions, this paper presents a tool that allows the analyst to point out these regions directly in the neutral format CAD-model. The corresponding surface representation is consequently created and inserted in the correct position and orientation in the model. This tool will complement the existing tools for mid-surface extraction, thereby providing more complete software for pre-processing parts with mixed geometry. In short, this paper proposes a strategy for pre-processing this type of CAD-model for the purpose of building a finite element model.

## 2. BUILDING THE MODEL

When dealing with parts of the mixed type geometry described in the introduction, several strategies exist to create FEA-models from them. The most straightforward method is to use continuum (solid) elements in the whole geometry, either by automated tetrahedron filling or by partitioning the model into sweepable portions. The problem may, however, be of such a nature that the whole model cannot be represented directly using solid elements. This is due to the fact that the number of degrees of freedom would result in an unreasonably long CPU time required for solving the model. Here, the mainstream strategy is instead to partition out the thin-walled regions first so that these can be meshed using shell elements. The remaining parts of the geometry can be meshed using solid elements. The two different types of meshes must thereafter be connected to a complete mesh.

In order to build an even lighter model and avoid spending time making the connections between the solid and shell elements, it is sometimes desirable to find shell representations of the non-thin regions in the geometry that approximate the mass and stiffness of these regions. This shell representation must be compared to the solid geometries they are intended to represent in order to see if the mass and stiffness of the geometry is represented in a sufficiently accurate way. A common method to check this is to solve the lowest free vibration frequencies either for the hybrid of shell and solid elements or just solid elements with one that only uses shell elements. If the frequencies agree, then it can be assumed that the shells represent the original geometry well in terms of mass and stiffness.

The instigator of this research has employed the latter strategy. Since the products are mostly die-cast and of a mixed geometry nature, this paper aims to propose a method of working that can

significantly reduce the time spent on pre-processing the models. The idea presented is to first use the mid-surface extraction algorithms provided in the FEA software. In this first step, the thin-walled geometry will be represented as mid-surfaces. However, this will not take care of the non-thin geometry. For this purpose, a complement to the mid-surface extractor has been developed. Namely, a program integrated in the CAD-system that allows the analyst to point out geometrical features that can be modelled as standardised shell representations that known from corporate experience to have good performance in approximating these typical geometries. The knowledge of what types of shell representation that is accurate for certain kinds of geometries is gathered from corporate experience working with FEA on similar types of products. Naturally, there will always remain some geometry that is not of any standardised type. Here, solid elements should be used and connected to the shell in the way earlier mentioned. The method can be summarized in the following steps:

1. Let the mid-surface extractor in the pre-processor create mid-surfaces where applicable.
2. Trim the created mid-surfaces to one connected surface representing all thin-walled geometry.
3. Use the CAD-integrated program to insert appropriate surface representations where possible.
4. Should there remain regions in the geometry that cannot be represented with shell element mesh, those are to be modelled using solid elements.
5. Mesh the surface representation with shell elements and connect the meshes to a complete FEA-model.

To elaborate on the details of the proposed way of working, the remainder of this paper is organised as follows: in Chapter 3, six typical geometries are processed in two different mid-surface extractors. This is done to find how much of the geometry that can be partitioned out as thin-walled automatically. How much manual work is needed to finalize a complete mid-surface for the thin-walled parts of the geometry? The CAD-integrated tool is further presented in Chapter 4. Finally, the conclusions of the work are summarized in Chapter 5.

### 3. AUTOMATIC MID-SURFACE EXTRACTION

There exist several algorithms for mid-surface extraction. One is “surface pairing,” as described by in (Rezayat 1996). Here, the algorithm seeks to find opposite surfaces and create the mid-surface. This algorithm laid the foundation for the mid-surface extractor in the CAE program I-DEAS™. Others have tried to make use of the medial axis transform (MAT) (Armstrong, Donaghy et al. 1996). In all cases, the surfaces need to be trimmed against each other to form a complete connected surface representation. This is problematic, since it has not been possible to formulate a theory that can deal with all types of connections between the individual surface patches. They have to be treated as individual cases, making it hard to guarantee success in every possible case.

The mid-surface extractors can be used fully automatically. In other words, the algorithm will go through the whole model and create mid-surfaces without intervention from the user. Another option is to manually pair the surfaces together by letting the user point at the two sides in the surface pair that will form the mid-surface.

The performance of two different mid-surface extractors is evaluated in the next section.

### 3.1 Automatically extracting the mid-surfaces.

As mentioned, one of the aims of this work is to find out to what extent the mixed models can be automatically processed in mid-surface extractors. Can the set of extracted mid-surfaces be used as a starting point in the continued work of building a complete meshable surface representation of the part? How much trimming of the created surfaces is needed?

Six different CAD-models were used. They are all tooling ready parts currently included in products on the market. They have all been represented with shell meshes, though not automatically created. The geometries are of varying complexity. The time spent manually creating the surface representations of these CAD-models is approximately one to three weeks for each part. Since the complete product consists of around half a dozen parts, automating the process or providing help is likely to have positive effects on the lead time. Furthermore, the results of the FEA will be available at an earlier stage, and that is a clear advantage in all product development.

The two programs used are HyperMesh® 7.0 from Altair Engineering (Altair 2008) and AI\*Environment™ 10.0.1 from ANSYS Inc. (Ansys 2008). No pre-processing of the geometries was done prior to starting the automatic mid-surface extraction. Both programs can import CATIA V4 models directly, so the models were saved in .model format in CATIA V5 and imported into the two FEA programs. The resulting mid-surfaces can be seen in the pictures in Figure. 1 below. The numbers displayed at each set of mid-surfaces are the number of individual surfaces created.

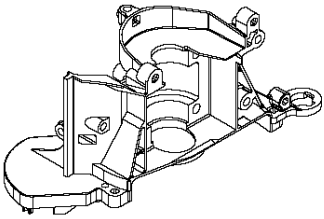
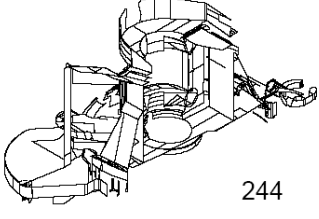
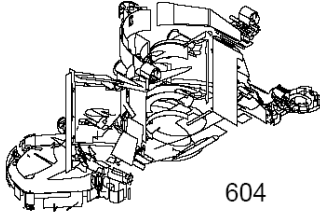
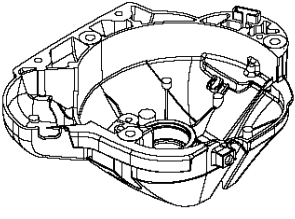
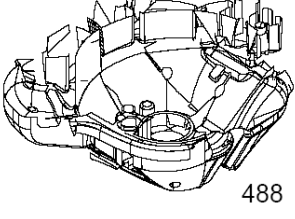
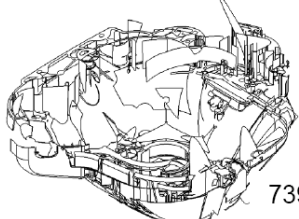
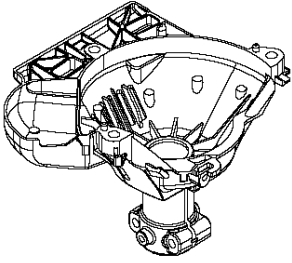
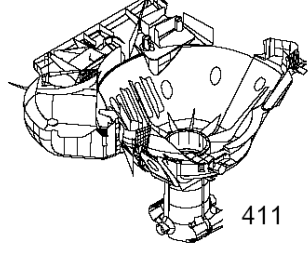
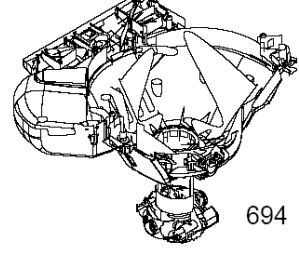
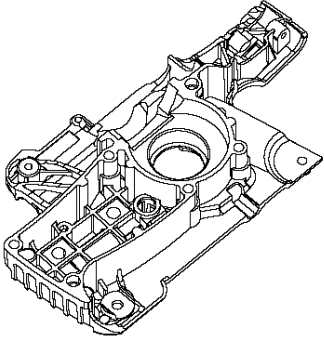
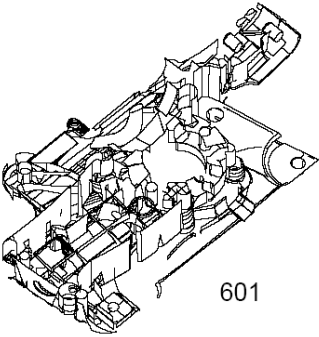
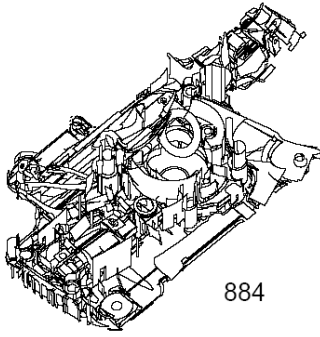
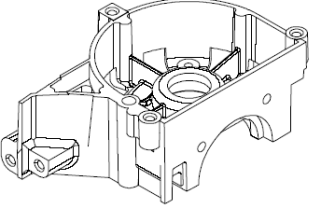
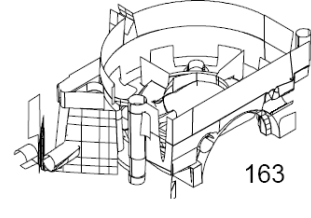
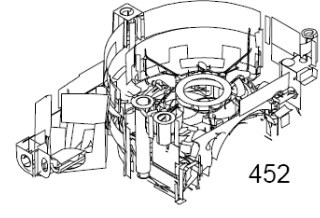
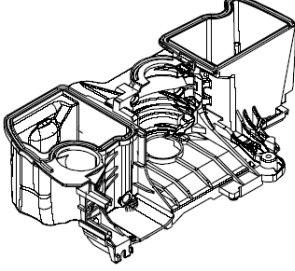
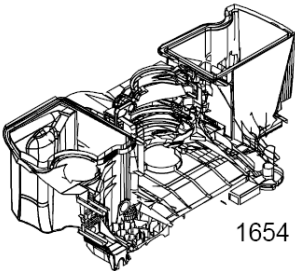
Original CAD-model	Hypermesh	AI*Environment
 1	 244	 604
 2	 488	 739
 3	 411	 694
 4	 601	 884
 5	 163	 452
 6	 1654	

Fig. 1: Mid-surfaces obtained using two different automatic mid-surface extractors.

As seen in the pictures, the two pre-processors seem to be accurate in finding the mid-surface between two boundary surfaces that are reasonably parallel. If the drafts on the surfaces are unsymmetrical or if the surfaces constituting the surface pair are dissimilar in any way, finding the mid-surface seems to be more problematic. The tolerance as regards dissimilar surface pairs seemed to be higher in hypermesh. AI\*Environment produced a higher number of individual surfaces, and some of the surfaces representing the ribs of the second and third model displayed a peculiar warped appearance. However, this may possibly be due to geometrical errors in the model. Furthermore, both mid-surface extractors created a large number of individual surfaces. This was expected since the search distance between the pairs of boundary surfaces was set well above the material thickness of the part in order not to miss any surface pairs. The number of individual surface patches could probably have been limited if automatic de-featuring (Armstrong, Donaghy et al. 1996) had been used prior to extracting the mid-surfaces. When this was tried, it seemed like the geometry was too complex. None of the software managed to remove more than a small number of features. As a result, the geometries were used in the condition they were provided by the initiator of this research. For part 6, it was not possible to create the mid-surface representation in AI\*Environment on the 2.2 GHz 1Gb Compaq laptop computer used. The CPU-time became very long.

The time spent on the subsequent trimming of the individual surfaces into one connected meshable surface was considerable. It was nearly equivalent to the time it would take to create the same surfaces manually. This is due to the fact that a large number of individual surfaces have to be stitched together. For this purpose, CATIA V5 was used. The resulting trimmed surface representation can be viewed in Figure 2 below. The trimming started by removing all the small surfaces the algorithms had identified around, for example, the small fillets due to the absence of prior de-featuring. To aid in this process, a script in CATIA V5 was created to scan through the created mid-surfaces and automatically remove surfaces with an area below a given threshold value.

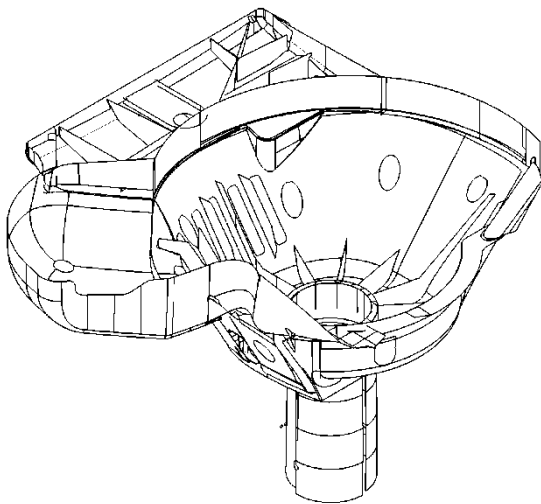


Fig: 2: Connected mid-surface representation.

After the automatic mid-surface extraction and subsequent trimming and stitching of the mid-surface representation of the thin-walled regions had been completed, representing the non-thin parts of the geometry with reasonable shell representations remained. In the next chapter, a prototype system for aiding the analyst in this task is presented.



#### 4. COMPLETING THE FEA MODEL

As seen in Figure 2, there are no surfaces to represent the non-thin parts of the geometry. In order to get them in place, the method that has been used for the creation of the tool presented in this paper is to study former designs and the surface/shell element mesh representation of them. This is done to build up a repository of features that occur in several of the parts. If they have been represented with the same type of surface in different design, then this particular representation is added to the repository. Examples of such features are stiffening ribs, screw attachments and features to distribute the forces exerted by the ejection pin assembly (here called “towers”). The same idea has been used earlier in papers on the same research (Paper I, Paper II). In these papers, it was shown how to use the construction history tree of the solid CAD-model to automatically identify these standardised features and to insert surfaces of the correct shape, size and orientation to represent the corresponding solid features. The system utilizes the fact that the CAD-models all have been modelled according to a corporate modelling instruction allowing the system to identify its features by searching the construction history tree. However, the system allowed only a few simple modelling commands to be identified, and this was insufficient for the complex models that had to be dealt with. It would have taken a great deal of effort to incorporate the sufficient number of commands so that this method could be used in the real industrial case. Furthermore, the parameterization of the parts shown in this paper does not describe the part directly, but rather the tooling used to produce them. A model of the ready part is created from a CAD boolean operation where the tooling halves are used to remove material from a volume. As a result, it was decided to instead use the model represented as a neutral format file. The idea of recognizing the features automatically from the construction history tree will naturally not work on a neutral format CAD-model. Instead, the user is asked to point out the geometric features in the model that can be represented as surfaces. The program will then automatically insert the surface representations in the correct orientation and trim them together with the surrounding surfaces.

##### 4.1 The tool for creating surface representations

The demonstrator has been programmed in VBA using the macro interface provided in CATIA V5. This allows the easy access of geometrical features in the neutral format CAD-model (such as the global coordinates for points located on edges). Since this information can be accessed, it provides an option to easily build the program. The user is asked to point at the edges in the model that constitute the features. The appearance of the main menu of the program can be viewed in Figure 3 below.

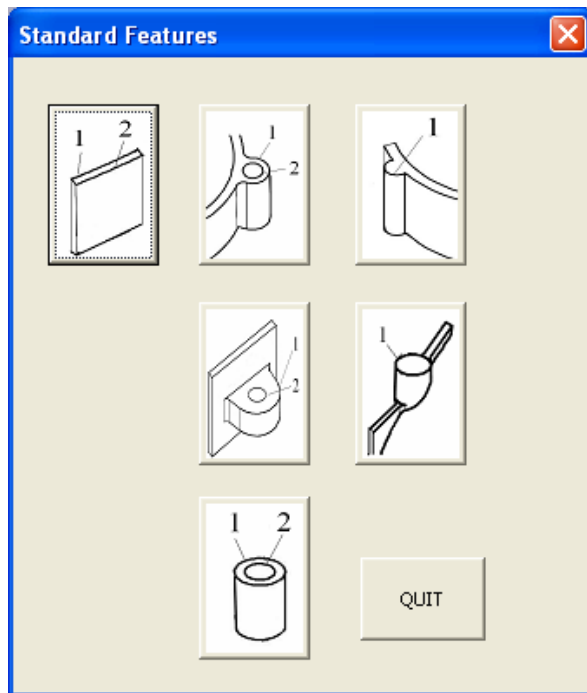


Fig: 3. Main menu of demonstrator program.

As seen in Figure. 3, there are three columns of buttons. Each column represents one family of standardised features, and each of the individual buttons represents one type of feature. The leftmost column represents ribs. The centre column has three different types of attachments, and the right column has two different types of ejection pin towers, as seen in the illustrations on the buttons. Should the user, for example, press the top button in the centre columns, the user will be asked to select the outer edge of the screw attachment in the CAD-model. It is in the illustration marked "1". This means that this edge should be selected first. Second, the user is asked to select the inner edge marked "2" in the illustration. After the selections have been made, the system will create a sketch on the same plane as the two edges are located, calculate the medial distance between the outer and inner edge and create an arc with a radius corresponding to the medial distance. Two lines tangent to the arc will be created starting at the arc's endpoints with a tangent constraint to the arc. The system will determine the location of the nearest surface and calculate the intersection between the lines and this neighbouring surface. These intersections form the endpoints of the two lines. When the sketch has been completed, the system inserts an extruded surface in a direction perpendicular to the plane in which the two selected edges are located. The surface is extruded to a distance indicated by the user.

Should the user have selected the rib tool in the leftmost column, the two edges that the user is required to select may describe either an arc or a line. The mid-segment between the two selected entities is created, and the ends are extended or foreshortened to intersect with the nearest surface feature. From the segment, a surface is extruded in the tooling draft direction or other direction specified by the user. The other four buttons work in similar ways. In Figure 4 below, the surface representation that corresponds to the buttons on the main menu can be seen.



Fig: 4. Surface representations that correspond to the standardised features of the main menu.

When the user has identified all standardised features in the model by selecting edges, the geometry is finally trimmed to make a complete surface representation ready for meshing. Figure 5 below shows the part after insertion of appropriate standard features.

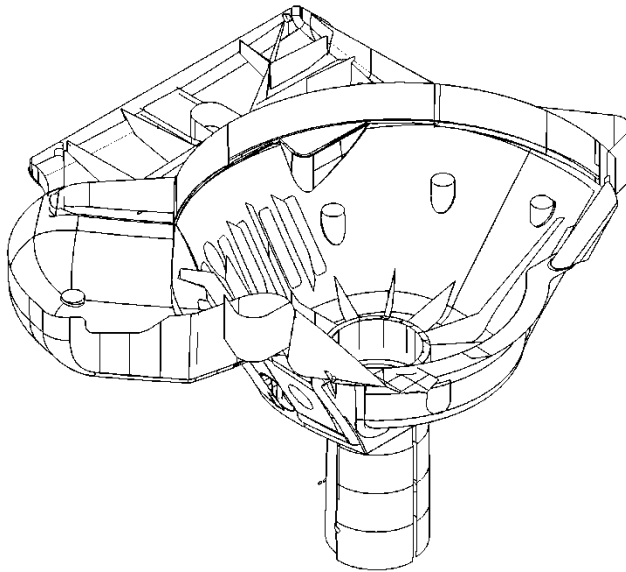


Fig: 5. Complete mid-surface representation.

#### 4.2 Finalizing the FEA model

Some regions remain in the model where it has not been possible to insert any standardized surface representations. Examples of such regions include the attachments on the “shaft” of the model (see Figure. 6). Here the strategy is to either mesh using solid elements (connecting it to the shell element mesh) or add a new form of surface representation to the repository after the careful evaluation of its performance.

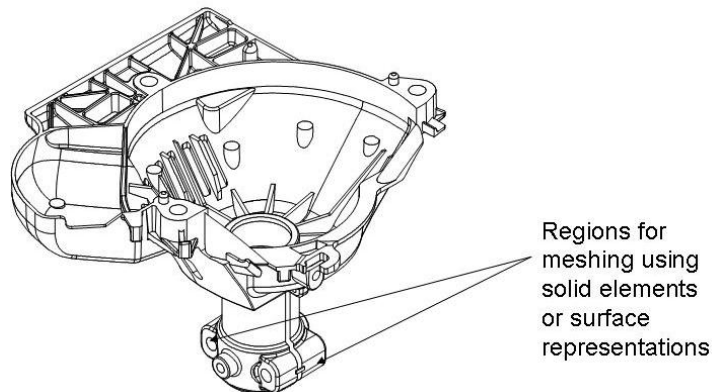


Fig. 6. Regions left after processing the model.

A FE-model of an individual part has been created ready to be assembled with the other parts in the FE-model of the complete product.

## 5. CONCLUSIONS

From the tests presented in this paper, it is clear that the mid-surface extraction algorithms in the FEA pre-processor are not ideal to use on the types of geometries at hand. They provide some help in partitioning out the thin-walled parts of the geometry, although they appear to be sensitive to small differences in shape in the surface pair. Both the number of individual surfaces that had to be trimmed together and the many small surfaces around, for example, the fillets could have been reduced if the prior de-featuring had been successful.

Finally, the time spent on completing the representation is about the same as creating the mid-surfaces manually. The standard feature representation tool presented will reduce the time spent creating these representations. However, here, time instead has to be spent on creating, evaluating, and maintaining the repository of surface representations. The best way to find the future best practice strategy is to compare the different ways of creating the FE-model by measuring the time spent on pre-processing the models in future projects. This should give a good picture of the performance of the proposed method.

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# IDENTIFYING FEATURES IN CAD-MODELS FOR POWDER METALLURGY COMPONENT EVALUATION

ROLAND STOLT

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## ABSTRACT

This paper presents a newly developed CAD-integrated system for the manufacturability evaluation of designs of powder metallurgy (PM) pressed and sintered parts. The contribution of the paper is the automated reconstruction of a specialized construction history tree from any CAD-model directly in the receiving CAD-system. The reconstruction is based on the geometrical restrictions of the shapes that can be manufactured by the PM process. This facilitates the creation of a transparent and user-revisable rule-base to evaluate the parts manufacturability, which is shown. It will enable designers to get feedback on their designs, reducing the number of design loops with the PM-parts supplier needed before the parts geometry can be established.

Keywords: Feature, CAD, Rule

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## 1. INTRODUCTION

Designing parts to be produced using the powder metallurgy (PM) pressing and sintering process requires detailed knowledge about the process. That is because the product and the manufacturing process are closely related. The parts geometry cannot be established without knowing the limitations of the pressing equipment and the manufacturing process in general. Since designers do not always have sufficiently elaborate knowledge, the aim of this research is to explore how a tool should be constituted to aid those designers in designing the PM parts so that the production issues are taken into account. The assumption is that the designer has some rudimentary PM manufacturing knowledge but is not familiar with the details. The designer is supposedly able to create a design suggestion in CAD, and the idea is to provide feedback on the manufacturability of the suggestion by evaluating the geometry against rules and recommendations for pressed and sintered PM parts.

In order to analyse the geometry directly, its purpose has to be known, giving it an engineering meaning. A common, well-known way of providing such meaning involves subdividing the model into features (portions of the geometry that have meaning, such as hole, slot, thread or pocket). Each feature can then be evaluated from various aspects. The features of a CAD model can be defined either directly, while defining the geometry (design by features) or after the geometry has been defined, by letting the user point out the features in the geometry. The features can also be retrieved automatically in a process known as feature recognition. (Shah and Mäntylä 1995) give an elaborate view on features and feature

recognition. Feature recognition is mostly used in machining applications, where it can be integrated into CAD-systems and used to plan machining operations (Nasr and Kamrani 2006). Other purposes have also been explored. One example is the decomposition of the geometry into features when automatically meshing the geometry for FEA (Dabke, Prabhakar et al. 1994). Naturally, features do not have to be limited to only machined features. Processes such as injection moulding also contain characteristic shapes that can be identified by feature recognition.

The automated evaluation of neutral format CAD-models for their manufacturability has been explored in the recent PhD thesis of Helen Lockett (Lockett 2005). She has developed a feature recogniser tailored for the injection moulding process. The aim is to make an advisory aid for designers of plastic parts. To recognise features such as ribs and bosses regularly appearing in plastic parts, she first extracts the mid-surface of the part. Removing one dimension will simplify the recognition process: one does not have to consider the third dimension when creating the rules that distinguish the different features. Topology graphs of the mid-surface are used to aid the rule-based feature recognition process. After identifying the features, an evaluation is made to determine how well-suited the identified features are for manufacture. The rules used for the evaluation are mostly based on experience and heuristics and have been gathered from suppliers of plastic materials and tooling technology.

For the PM pressing and sintering process, (Smith 2003) has customized a CAD-system. It allows the designer to select among a number of standardized features that can all be produced by the PM process in a design-by-features approach. The system, called IDA (Interactive Design Advisor), works by building the model in the CAD-system by combining such features. When a feature is added to the design, a rule-based check is made to secure that the dimensions of the feature (set by the user) and the combination of features are feasible from a manufacturing point of view.

These two systems both make a manufacturability evaluation based on heuristic rules and recommendations. The rules are general and consider all types of products and materials. Consequently, the precision in these rules cannot be particularly high. The rules need to be refined to consider the specific equipment and materials, as well as the particular type of products being designed. All this has to be reflected in the system. Therefore, there clearly is a need for the users of the system to be able to revise, add or delete rules. One way of letting the content and number of rules vary but still keeping them computer executable is to use rule-based systems see e.g. (Hopgood 2001). Rule-based systems that involve geometry are sometimes referred to as Knowledge-based engineering systems, KBES. Nowadays, facilities for defining such systems are sometimes integrated into the CAD-systems allowing the definition of rules and checks to evaluate the geometry. Further, the CAD programs may also allow the automation and user programmable access to the functions in the CAD-system. The work presented here has been carried out in such a CAD-system complemented with the necessary application programs.

As previously mentioned, in order to create the KBES rules and checks, the geometry and topology needs to be given an engineering meaning so that the KBES can interpret it. Therefore, a key issue for the work presented here is if it is possible to establish a modelling procedure using the features provided by the CAD-system (such as extrude, revolve, and loft) such that all or most of the shapes that can be manufactured can be modelled in a standardised way. The structuring and parameterisation of the model in a predefined way will mean that the geometrical elements and parameters sought can easily be retrieved and evaluated by the KBES.

In order to keep the number of features and feature combinations on a manageable level so that the KBES will not be overly complicated, the number of permissible features needs to be limited. However, this is not necessarily preferred from the designer's point of view. The CAD developer has put much time and effort into developing modelling commands for the efficient definition of the geometry. Further, the designer has often developed a personal proficiency in using them. Instead of restricting the modelling procedure, the intent is to automatically construct the desired construction history afterwards by evaluating the low level entities in the model: edges, faces and vertices. Doing so is known to be difficult involving feature recognition. Nonetheless it is feasible, given the strict geometrical restrictions of the process. That is, at least if some simplifications are made and those will be shown in this paper. This also has the added advantage that neutral files (such as STEP or Iges) originating from arbitrary CAD- system can also be processed. It should be noted that construction histories may become interoperable between CAD-systems in the future. However this is not presently possible (Kim, Pratt et al. 2008).

To summarise, the aim of the work is to explore how a system to prepare CAD-models for the geometric evaluation of their manufacturability should be built. The actual evaluation follows the industrial practice by defining rules and checks in the construction history tree of the parts. The rules will then be transparent and accessible for the user, allowing the contained knowledge to be refined (rather than hard-coding it into the system). Consequently, the contribution of the paper lies mostly in the preparation of the models (in other words, finding a standardized way of modelling all pressed and sintered PM parts and automatically reconstructing the predefined construction history from arbitrary CAD-models). The reason for adopting this unexplored approach is primarily the user refinement of the knowledge. However, it is also to be independent of modeller and modelling procedure. Further, it enables the automated evaluation of a large number of existing PM-parts, which perhaps will contribute to refining the general rules and recommendations of PM-parts.

The industrial benefit of the system is allowing designers to evaluate their design suggestions so that mistakes hopefully can be avoided. This will increase the process knowledge of the designers, thereby facilitating the communication with the PM part suppliers and reducing the number of redesign loops required before the parts geometry can finally be established.

## 2. METHODS AND TOOLS USED

A number of parts produced by PM pressing and sintering have been studied. The modelling method is established by modelling the parts in the commercial CAD-system Catia V5 (Dassault-Systemes 2008) using a limited number of CAD-features. This establishes a modelling procedure by which the geometries can be defined using primarily linearly extruded convex and concave features.

A simple feature recognition technique is developed to find the PM features in neutral files. The feasibility of the method is tested by creating an automated experimental system using the application programmable interface (API) of the CAD-system, complemented with software programmed in visual basic. A number of in-production PM parts are processed in the system to verify the accuracy of feature recognition and geometry evaluation.

## 3. DERIVING A CONSTRUCTION HISTORY OF PM PARTS

A brief description of the PM pressing and sintering process is given to aid the understanding of the feature structure and feature recognition procedure. The feature recognition procedure is described, as well as the construction of the parameterised model, in the receiving CAD-



system. It is also shown how the parameterised model is evaluated for its manufacturability using a rule-based system defined in the CAD-system.

### 3.1 The PM manufacturing process

The PM pressing and sintering process is limited in terms of what shapes can be manufactured. The primary reason is that in order to press the powder to an even density, the compression rate must be equal everywhere in the part. This is mainly because, unlike fluids, powder has very little possibility to redistribute itself inside the die during pressing due to the high internal friction in the powder and against the die walls. Consequently, the part must be designed with a limited number of levels in the pressing direction, each formed by a separate punch in the tooling assembly. Minor features can be formed by indenting the faces of the punches, called face-forms. They will inevitably introduce a distortion in the density distribution and should be avoided, especially in areas where high mechanical properties are needed. The reason for wanting a high and evenly distributed density is that the residual pores between the powder grains have a very detrimental effect on the mechanical properties of the final part, see e.g. (Beiss and Dalgic 2001). Consequently, when designing PM parts used as loaded structural elements, care should be taken to place low density areas where the part is least stressed.

Following from the above, the freedom of shape and the accuracy of tolerances are quite low in the pressing direction ( $\sim$ IT13). Meanwhile, in the radial direction, complex shapes with high dimensional accuracy ( $\sim$ IT9) can be formed. Further, to avoid making an overly complicated and expensive tooling, the number of punches must be limited to typically two upper and three lower.

Figure 1 on the next page shows a part pressed in a tool with two lower and one upper punch. Note that the outer lower punch could possibly have been replaced by a step in the die. The punches must move separately, creating a uniform pressure distribution in the powder. This is realized by attaching the tooling elements to plates sliding on rods. The plates are moved by hydraulic actuators. The individual movements of the actuators are, in newer presses, guided by CNC.

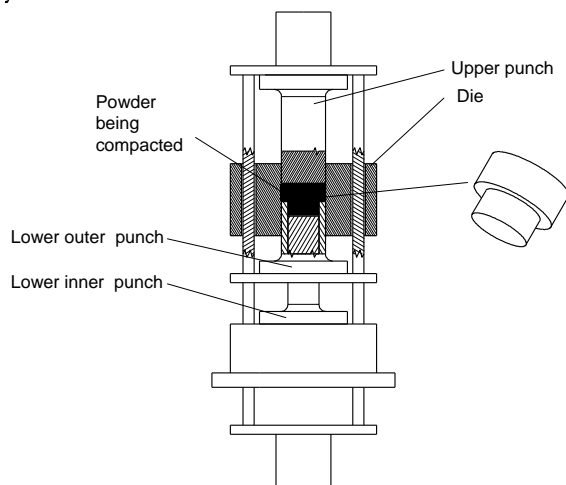


Figure 1. Compaction tooling assembly.

### 3.2 Feature recognition

The features in the neutral file CAD model need to be found to derive a construction history in the receiving CAD-system. In order to do so, a simple feature recognition procedure has been developed. Feature recognition is, as mentioned, a well-established area that have been studied for a long time. It can be widely simplified in the case of PM pressed and sintered parts and for the purpose needed here. The PM parts are dominated by one type of feature; namely, what is here called a powder column. A powder column has a uniform section and represents one pressing height. A PM-part can in many cases be seen as a number of such columns stacked or placed side by side. Thus, the feature recognition process only has to find the flat areas, the levels, perpendicular to the pressing direction marking the ends of the powder columns. This is done by searching the neutral model for edges that form loops located in planes perpendicular to the pressing direction. When such a loop is found, parameterised lines and arcs are fitted to it. The reconstructed loop can subsequently be extruded in the pressing direction to form a solid feature with the same height as the original. An example to illustrate the procedure is given in the next section. The PM parts may also contain other features such as drafted sections and chamfers and rounds located on planes perpendicular to the pressing direction. It may also contain minor features that have been formed by face-forms. These have been disregarded for the time being.

### 3.3 Creating a construction history tree and evaluating the part

In the neutral CAD-file, the loops of edges forming the flat faces can be distinguished from the model by extracting edges that form loops in the model staying at a constant level in the parts pressing direction. What side of the loop that the material in the model is located is also determined. Figure 2 on the next page shows a part that has edges forming ten closed loops in the pressing direction. Parameterised sketches are formed in the receiving CAD-system by fitting lines and arcs to these edges (1). The closed loop sketches are linearly extruded to a distance corresponding to the full height of the part in the pressing direction (2). Since it is known which side of the loop the material is located, the loops located between the end levels are used to remove material from the model in the opposite material direction (3). Now a part with a construction history tree and parameterised sketches has been constructed in the receiving CAD-system.

The observation that PM parts can be constructed by sketching sections that are extruded has been explored earlier. In the PhD work of (Dissinger 1995), an experimental system is presented where the user sketches the sections. A rule-based evaluation of the manufacturability of the sketched sections and the interactions between them is made prior to extruding it into a solid.

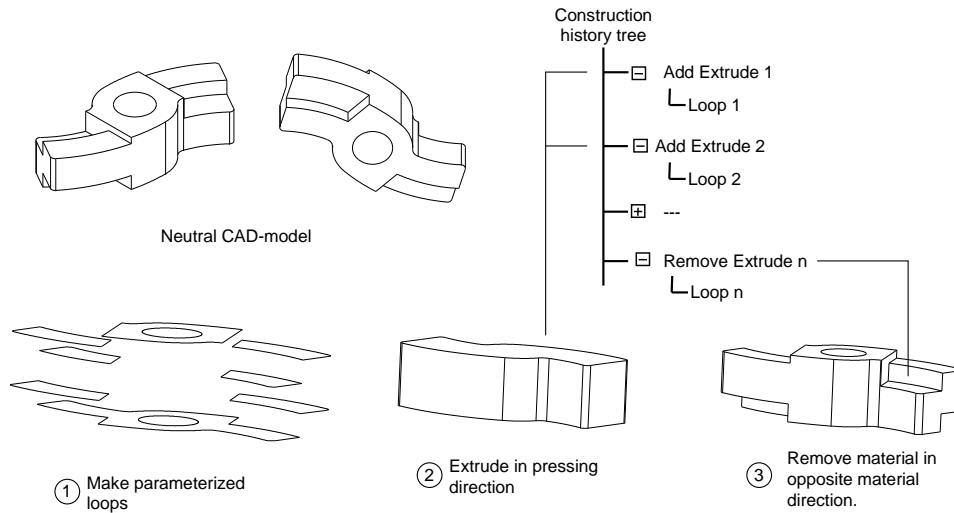


Figure 2. Deriving the construction history.

### 3.3.1. Limitations of the method

Having a levelled part with the levels perpendicular to the pressing direction is a prerequisite for the proposed recognition process. It is also ideal from a manufacturing point of view. However, there are examples of PM parts that do not fulfil this requirement. These parts often require specialised tooling technology and are therefore not very common. The Figure 3 below shows two examples of parts that will cause problems. In the left part the levelling is not perpendicular to the pressing direction in the protruding “wing” features. Consequently, they are not recognised. The wedge shaped part on the right cannot be processed for the same reason either. A more elaborate discussion of the principles of the underlying tooling technology for compacting parts like these examples is given in (Beiss 2007).

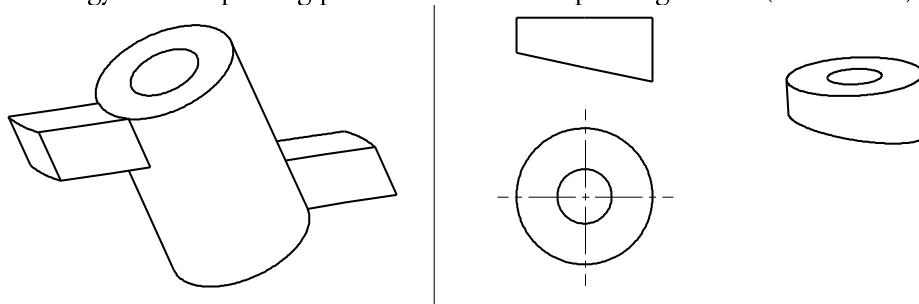


Figure 3. Exception parts.

However, having examined a number of in production parts, it seems like most PM parts can be processed. This is supported in Höganäs handbook, Chapter 8 (Höganäs 2007). There, 35 examples of PM parts from 9 different difficulty categories are given. Among these only 9 contained spherical, sloping and others that cannot be identified by the prototype system. Note that most parts contain chamfers and rounds that were disregarded. The successful modelling of the parts using simple features indicated that the PM process is well-suited for the proposed method. Other processes, such as machining, can create complex feature interactions, making the individual features difficult to identify.

### 3.3.2 Evaluating the parameterised model

The derived construction history tree is far from being practical if the geometry was to be defined interactively by a designer. The designer would perhaps have used the part's antisymmetry and created only half the part, constructing the other half by copying and rotating the copy half a turn. However, the parameterisation has provided a structuring of the geometrical elements of the model that have made them available for manufacturability evaluation using a rule-based system. The rule-base to evaluate the part contains a number of geometrical recommendations on how the PM part should be designed to enable the creation of a reliable tooling. These recommendations are found in handbooks such as (Mosca 1984; MPIF 1998) and are based on experience from PM parts production. The recommendations are aimed at insuring that a reliable tooling that can compact the powder to as uniform density as possible can be manufactured. In order to check that the parts comply with the recommendations, a number of checks have been defined in the CAD-system. The CAD-system supports the definition of such checks through an integrated KBES shell. The checks are formulated as production rules of the type IF <Condition> THEN <check pass> ELSE <check fail>. A few checks are shown in Figure 4 below. It is, for example, checked that there are no angles in the geometry that are less than  $30^\circ$ . If there are, the tooling elements that form the edge will become fragile and probably break in the production. Further, it is checked that there are radii at the inner corners of the loops to facilitate the flow of powder in the tooling. Whether the tooling elements become too thin is also checked. There is a risk that they deform unacceptably, due primarily to the high compacting pressures. The compacted, but not sintered, part is quite delicate and can easily break as it is being ejected from the tool. Therefore, it must not have any long protruding sections that are susceptible to fracture. Due to the high friction within the powder and against the die walls, the ratio between the diameter and the height in each section must be kept within limits. Otherwise, the density will become too low in the area furthest away from the punch faces. This condition is also checked.

The structuring of the geometry mean that data can be collected from the model through the CAD-system's API. The data relevant for each check is collected. One example is when determining if the parts have any small angle edges. First, all angles between the sketch-elements defining the loops are measured. From these measurements, only angles less than  $30^\circ$  are sorted out. If such angles are found, the check fails, the user is shown where the small angles are found in the geometry and the user is prompted to revise the geometry until it passes the check.

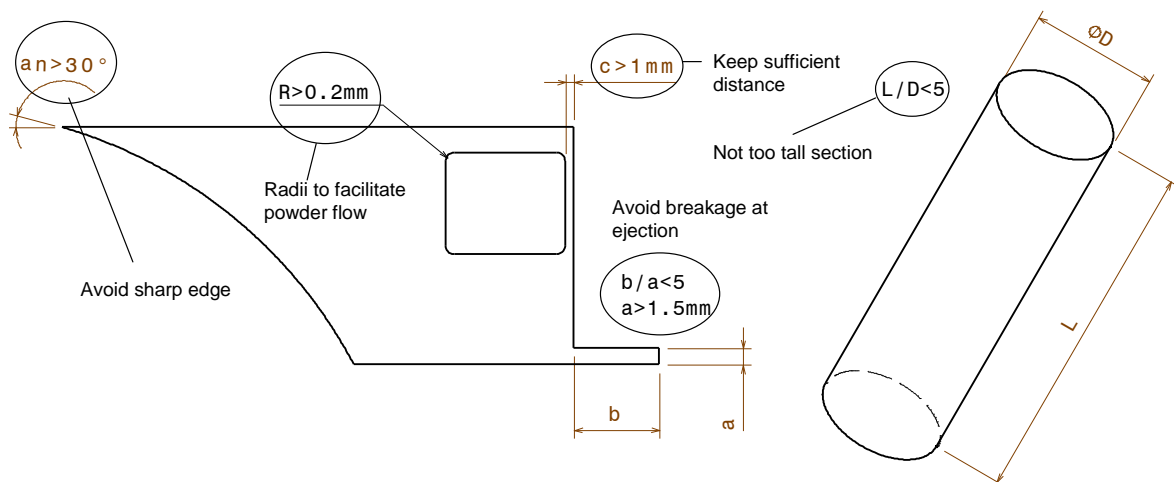


Figure 4. Geometrical conditions checked.

This process continues until all checks are passed, meaning that the part complies with the geometrical recommendations for PM parts. The reconstruction of the part in the receiving CAD-system means that the check becomes transparent to the user. If additional checks are needed, then the elements of the model are easily accessed through the API and evaluated by additional rules. What has been evaluated also becomes apparent to the user since all sketches, sketch elements and positive and negative extrusions are available in the construction history tree of the part.

Figure 5 shows three of the PM parts that have been evaluated in the system. They pass all checks, except the rightmost part. It was found to have a too small angle at the location indicated.

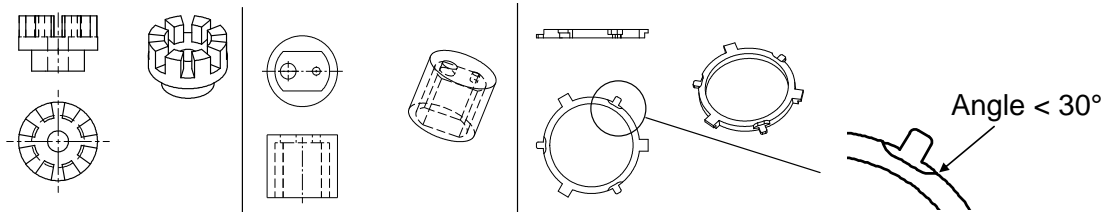


Figure 5. A selection of the PM parts tested.

### 3.4 Possible uses of the described system

The system is intended as an advisory aid for designers of PM parts. It has the added advantage of allowing designers to incorporate their gathered knowledge in it. As an alternative, it could be run on a web site to which the designers could upload their CAD-models and get recommendations on how to revise the geometry to make it more favourable from a manufacturing point of view. Further, it will encourage the designer to start thinking in terms of tooling technology. Failure of the system to identify the features means that they deviate from the ideal levelling of the part. In that scenario, perhaps a design revision can be made to make the part more favourable from the manufacturing perspective. This will lead to an increased awareness of the pros and cons of the manufacturing process, enabling the designers to be more confident when communicating with the PM-part supplier.

## 4. CONCLUSIONS AND FUTURE WORK

It is concluded that parts produced by the PM pressing and sintering process, as a result of having strict shape limitations in the pressing direction, lend themselves to being modelled using a limited number of simple features. These features can automatically be extracted from neutral format CAD-models by identifying the individual levels in the part that is planned to be formed by the tooling elements. Reconstructing the identified features in a commercial CAD-system will mean that the creation of a transparent and user-revisable a rule-base for manufacturability evaluation is facilitated.

### 4.1 Future work

For the work presented, it is assumed that most pressed and sintered parts can be modelled by simple extrusions. This seems to be a reasonable assumption, since it is a prerequisite for the ideal levelling of the part. Nevertheless, real parts also display other features: face-forms, chamfers, rounds and so on. The system must be further developed to recognise and evaluate these. One example is that chamfers and rounds should not be located so that the tooling elements need to be manufactured with sharp edges that are quickly worn out.

Here material suppliers and PM parts producers that participate in this research project can provide valuable help. They can provide both the necessary test environment and examples of PM parts. The expectation is also that this will lead to the further development and refinement of the rules and recommendations for pressed and sintered PM-parts.

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# AN AUTOMATED DESIGN AND ADVISORY SYSTEM FOR PRESSED AND SINTERED POWDER METALLURGY PARTS

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## ABSTRACT

The aim of the present paper is to dissolve the bottleneck for PM market expansion caused by lack of knowledge among designers about PM-specific design principles by proposing an automated PM-part advisory system. The system, called PM-Wizard, checks that a PM-part design suggestion represented as any 3D solid CAD-model complies with the geometrical design rules and recommendations for PM-parts. The system consists of a rule-base hosted by a modern commercial CAD/CAE-system and a programmed algorithmic procedure that converts the design suggestion into a specially developed format. The format enables the automated evaluation of the PM-part in the receiving CAD/CAE-system informing the designer on how the geometry should be revised to facilitate the pressing. The conversion algorithm has been tested on a large number of PM-parts, and the result is that all the tested parts can be converted into the proposed format and successfully evaluated.

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## 1. INTRODUCTION

Producing parts by powder metallurgy (PM) pressing and sintering can have several advantages compared to other production methods. Good material utilization and producing a net-shape with good dimensional tolerances are a few examples of attractions. Although the PM process in some cases is a good choice of production method, this may not be obvious to the designer working in product development (PD). The designer employed at, say, a car manufacturer has to realize that PM is a feasible production method for a certain component. He or she has to be reasonably familiar with the geometrical restrictions of what can be produced by PM. Further, the designer has to have a rough idea as to whether the PM part can fulfil the specification for the product. For example, can the required wear and fatigue resistance be obtained?

To make a rational choice of production method, the alternatives have to be compared. This requires accurate and up-to-date knowledge of the capabilities of the alternative processes. Once a part has been identified as a candidate for production in PM, the designer works out a preliminary design. This design has to fulfil the geometrical, mechanical and other requirements made clear in the product specification. With the preliminary design

established, it is likely that PM suppliers are contacted to help finalize the envisaged PM part and come up with a quotation for the anticipated cost. Several loops of redesign of the part are likely to be needed before agreeing on a final design that can be economically produced and, at the same time meets all the requirements. Making the preliminary design may be a difficult task for the designer. He or she may have basic knowledge of the PM process and some experience working with the development of PM parts. Nonetheless, the designer cannot be expected to know all the details of the PM process, especially since it is a complex process with many processing steps and a multitude of parameters all affecting the end result.

When making the preliminary design the designer presumably will turn to handbooks such as (Mosca 1984; MPIF 1998) that contain guidelines and recommendations regarding how to make the part best suited for production in PM. There are also resources available on the internet such as (PMProperty-Database 2007) to assist the designer in selecting the powder. Since the designer possesses knowledge about the product and its specification and not the details of the production method, it is important to involve the PM supplier at an early stage in the PD process. In that manner, as many potential errors as possible can be avoided. However, given the complexity of the process there are variations between different suppliers. The designer should not take for granted that the opinion of a single supplier is representative for the whole PM business. One supplier may say that a particular shape is not possible to compact while it is common practice for another. It all depends on the available equipment and what types of products the supplier is familiar with. Furthermore the PM process, as well the alternative processes, compete for the same products. The production processes are constantly being developed, expanding their capabilities. It is now possible to produce PM parts subjected to fatigue and wear, such as synchronizing rings, connection rod and spherical gears. The emergence of CNC presses that give very accurate control over the tooling motions has largely contributed to this development. In (Beiss 2007) a number of innovative tooling design are described.

As a consequence of the advancements in PM technology, products that have a potential of being profitable if produced by PM run the risk of being rejected at an early stage in the PD process due to insufficient and obsolete knowledge of the capabilities of the process.

How should this problem be tackled? It seems like computer support is required for acquiring, maintaining and applying the engineering knowledge in the correct situation. The theoretical foundation behind such computer support is well-established and has originated from research in computer science and knowledge-based systems (KBS). These tools have successfully been applied in the context of engineering, as described in many textbooks, such as (Krishnamoorthy and Rajeev 1996; Sriram 1997; Hopgood 2001).

### **1.1 AI used in Powder Metallurgy**

To date, there are few, if any, reports of systems that operate in real commercial PD of PM parts. In research reports, however, there are many interesting references to be found. Reports of automated PM design systems with geometrical capabilities date back to the 1980s (Dunbar and Bradley 1984) but it seems as though the study of these systems gained momentum in the 1990s. In (Zenger, Kim et al. 1995), a system for the automated design of tooling is presented. The intention is streamlining the process of designing the pressing tooling. The starting point is a finished PM part design including material and requirement specification. The part geometry as well as the textural information is entered into the system by allowing the tooling designer to select surfaces and construction lines in the CAD-model defining the boundaries of the model. Next, the principal setup of the tooling is established. The pressing direction, number of upper and lower punches the type of core-rod and die and so on are defined. This means deciding which tooling elements are going to form the different

geometrical features. With the principal tooling setup in place, the system can decide which of the company's presses is the most appropriate by matching the required stroke and press tonnage and so forth against a press database. The dimensions of the tooling elements are very seldom the same as the nominal dimensions of the part. Consequently, the dimensions of the tooling are compensated for material shrinkage in sintering, deflection of tooling and so on. These compensation factors are stored in a database and can be revised as the corporate knowledge increases. Now the tooling elements can be created allowing the designer to work out how they should be fitted into the adapter system of the selected press. The detailed design drawings of the tooling elements can now be made in interaction with the designer according to corporate drawing standard. The information the system relies on is kept in a total of nine different databases. Examples include material for part and tooling elements, tool geometry, presses, and tolerance and clearance.

In the PhD thesis (Dissinger 1995), a system for aiding the designer in creating PM-parts is presented. The user gets immediate feedback on the work by letting a rule-based system monitor the added items. The key idea is to let the user build the part by creating 2D profiles. These are sketched in a specialized sketching editor. The sketched 2D profiles are then extruded to 3D entities. This way of working is based on the assumption that, in order to press the powder, the parts need to be designed in a limited number of levels each formed by the elements in the press tooling assembly. Analyzing the 2D profiles is easier than analyzing the 3D entities directly because it is restricted to planes, the author argues. The system consists of a graphical user interface (GUI), allowing user control over a number of rules and functions derived from modeling and production considerations. The system evaluates the model by checking the rules in sequence and lets the user revise the model until it complies with all the rules.

The system is comprised of a total of 51 rules and functions where the majority is for calculation and checking conditions for modelling and production considerations. Five of them are functions that have been written to allow modification of the model (e.g. the addition of features needed for production reasons such as fillets, axial flats and tapers).

The functions employ mostly geometric reasoning in algorithms devised by the author. Examples of conditions that are checked are that the profiles do not self intersect or intersect each other, checking that the object can be created by the CAD-modeler. The majority of the rules secure the resulting geometry can be produced by PM, such as checking the profiles so they do not have too narrow sections (impaired powder filling). It also checks that the profiles are not located too close to each other (thin tooling elements). Furthermore, the profiles are checked so that they do not produce feathered edges. A feathered edge is when a contour in the part has an insufficient angle, so that the tooling element needs to be manufactured with a sharp and thereby fragile edge. This can also make the powder filling problematic. If there are notches present in the profiles, these are checked so that they do not have unacceptable length/width ratios. The violated conditions are added to a list at run-time, and the user is prompted to revise the model until no more violations remain.

In the work of Lyndon Smith (Smith 2003) an attempt is made to address the whole process of designing parts for production in PM. Smith proposes a KBS that, apart from aiding the designer in creating parts geometrically apt for the process, also addresses the material selection, making a prediction of the expected properties as well as the outcome of the sintering process. In short, the system addresses all aspects of the process. Understandably, such a system needs an extensive knowledge-base. Smith proposes to build up the knowledge-base through various techniques. First, for making a sound geometrical shape ensuring that the part can be pressed, Smith has developed an interactive design advisor (IDA) (Smith and Midha 1996). Through a GUI, the user is allowed to select standardized features, such as

cylinder, hole, gear, and chamfer. The user enters the dimensions of the selected features when adding them to the model. The feature dimensions, as well as the combinations of features are evaluated against a set of rules. As soon as a dimension or a combination of dimensions (possibly from more than one feature) is outside the permitted range, the applicable rule fire and the user is prompted to revise the design. This process continues until the design is completed without triggering any of the rules. The system is defined in AutoCad® and employs LISP programming.

The KBE proposed in (Smith 2003) uses mathematical functions have been fitted to experimental data through statistical analysis for the selection of powder. This provides the user with a means to make more accurate interpolation in the material data. Published material data are in most cases taken from standardized test samples. Real PM-parts are geometrically more complex than the test samples, so it is not certain that the resulting density everywhere in the part will be the same as in the test sample. Smith envisions that the knowledge on how the actual part geometry correlates to the tested samples should be gathered from numerical simulations of the compaction process to collect representative cases from which conclusions can be drawn in similar cases. Finally, the sintering process is addressed by entering the process parameters of a large number of runs into a neural network (NN). The motivation of proposing a NN is that the sintering process is complex. There is a lack of models including all relevant processing parameters.

The systems described are focused on different users. The systems proposed by Dissinger and Smith are indented for supporting the design of the actual parts, whereas Zenger focuses on the tooling designer. At a first glance, this may seem like different types of systems. However, since all design rules and recommendations for PM parts are aimed at securing that the parts can be produced according to the requirements, the product and its production process cannot be separated. In fact, due to the close relationship between the product and its process, it is not possible to make any predictions on the properties of the final part without knowing the principal setup of the tooling. One example is that the density of the compact can vary largely depending on whether a certain feature is formed by a separate punch or by a face form. The end face of the punch has been shaped to form the feature, thus the name face form. Likewise, whether it is single or double sided compaction has a decisive influence on the resulting density. If a more detailed prediction is required simulation will perhaps be employed. Even if the designer does not go to such lengths as making a compacting simulation, it is still necessary to know the basic tooling setup. Rules of thumb such as the following is likely to be used: the further from the punch face, the lower the density or the higher the face form is compared to the total height of the pressed column, the greater disturbance in the density distribution. Certainly, no absolute values can be predicted in this way. Still even for rough estimates, the planned compaction tooling setup needs to be known.

## **1.2 Requirements for an automated design system for PM parts**

Clearly, there have been several well-founded proposals regarding how to build systems to support the development process of PM parts. Yet still it does not seem like there is anything commercially available yet. The reason can possibly be found in the fact that both the production process and the products develop rapidly. There are some basics that are fairly constant, such as the ideal shape of the part to make it most favorable for pressing. The powder materials and tooling technology however, are constantly developing. Providing a system commercially would mean that great effort would have to be put into keeping the system up to date, especially since the users' demands vary. Also, the knowledge about the specific products, which the users' expertise is not practical to support centrally. All this can

be tackled by allowing the users to take an active part in the refinement of the system and the adaptation of it to their own products by using systems that support this way of working.

Lately the CAD, simulation, and KBE software has developed towards better integration, allowing the different software to operate on the same parametric CAD model in an integrated environment. Examples of such software are Dassault systemes CATIA Knowledgeware® and Siemens NX Knowledge fusion®. In addition to being efficient in defining the geometry, they are also capable of analyzing designs by allowing the user to define rules, checks and formula manipulating the parameters of the model. Further, they allow integration with simulation software code providing means of computationally evaluating and optimizing the designs. Provided that the system is equipped with an application programmable interface (API) allowing the functions of the CAD-system to be controlled by programs defined by the user. This creates a purposeful environment in which to host the proposed system. This paper seeks to explore how this can be utilized in the development of PM parts with a special focus on the transparency and the user maintainability of the knowledge of the system. Further, following from the above review, it is believed that an experimental system should concentrate on the following issues:

- Providing recommendations on design proposals regarding the geometry of the part, in order to make it favorable from a manufacturing point of view.
- The integration with commercial CAD software, allowing the user to take an active part in the refinement of knowledge used in the system.
- To visualize for the designer how the tooling setup to produce a proposed part would look. This will also make it possible to predict roughly the expected properties of the part.

For the study of the above questions, a prototype system for integration with the commercial software CATIA is being developed. This experimental system has been given the working title PM-Wizard.

## 2. PM-WIZARD

The intended users of PM-Wizard are design engineers with some basic knowledge of the process, but who are not familiar with the details. PM-Wizard is planned to enable these companies to gain an understanding of the process so they can be more proficient in the communication with the PM-parts supplier. Time will be saved by limiting the number of design cycles needed in corporation with the PM-parts supplier. It is also thought to aid the designer in selecting the appropriate supplier on account of being more aware of the pros and cons of the process. The system is also expected to help structuring, storing and refining the corporate knowledge on the product and process.

As a first step that is described in this paper, PM-Wizard converts any CAD-model to an interpretable representation and checks that the rules and recommendations for PM-part design have been followed. This is a first coarse check and employs the knowledge typically found in handbooks. These checks do not consider the compacting tooling design or the powder material used. Consequently, the precision is low. The system is designed so that, if the design recommendations are violated the user is prompted to revise the design until all checks are passed.

Since the pressing tooling plays an important role for the more detailed design, recommendations and the prediction of the expected properties is believed to require that the system have some basic type of automated tooling design to suggest alternative tooling setups

to the designer. Further, in the continuation, the powder material, the intended pressing equipment and the planned process route (such as secondary densifying operations and the sintering procedure) also need to be taken into account, since they are all decisive for the final result. Figure 1 below shows a preliminary layout of the envisioned system.

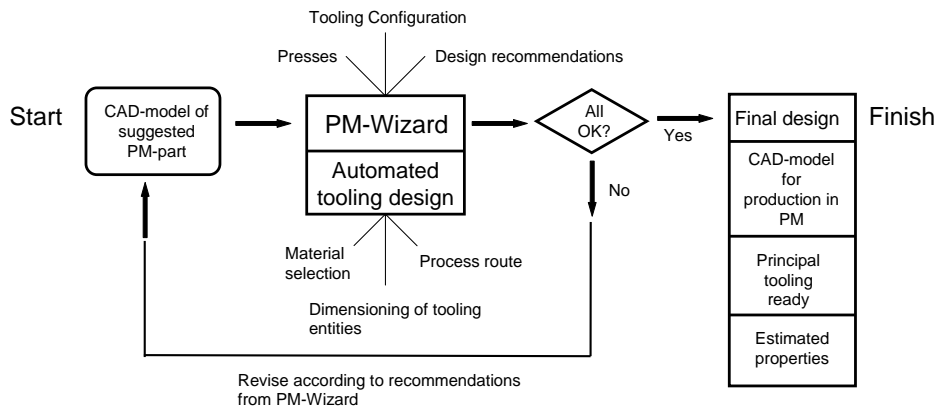


Figure 1. Planned function of PM-Wizard.

## 2.1 System development

PM-wizard has been partly integrated into a commercial CAD-system since the handling of geometry requires advanced software that would not be feasible to develop solely for this study. Further, the CAD-system also contains a shell for defining rule-based systems allowing the definition and managing of the rules. The rules can be explicitly read, understood and revised by the users of the system. The execution of the rules is controlled by a forward-chaining inferencing mechanism. Further, the system can be controlled by application programming so that the different functions provided can be seamlessly integrated.

## 2.2 Automated evaluation of the PM-part

In order to utilize this functionality in the CAD-system, the CAD-model must be defined in the CAD-system in such way that its parameters and construction history tree can be interpreted by the functions that have been devised and adapted for the purpose. Consequently, the designer must create the PM-part CAD-model in a predefined way so that the models features and parameters are structured in a predictable way. This may not always be practical. The designers will be very limited in the modeling techniques that they can apply. The modeling technique used is often a matter of personal preference. More importantly, access to the particular CAD-system and knowledge on how to use it is required. This would drastically reduce the scope CAD-models that can be processed.

To get around these problems, the idea is to create an automated routine that can read the geometry of the design suggestion defined in an arbitrary CAD-system and then reconstruct the geometry in the receiving CAD-system, now with the desired construction history tree and parameterization. Figure 2 below shows the intended process.

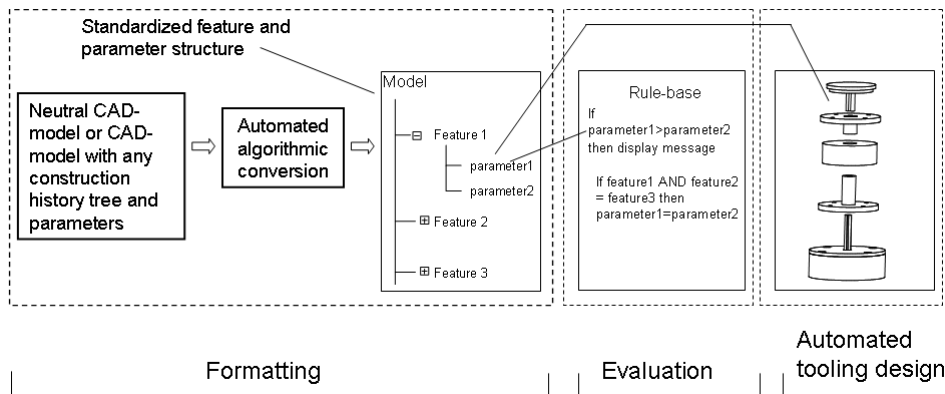


Figure 2. Algorithmic conversion of arbitrary CAD-model to PM format and proposed used.

Having the model represented in this standardized way will facilitate the manufacturability evaluation and the creation of automated design systems.

Finding the features is complicated in the general cases. It involves the automated recognition of features in the geometry (see e.g. (Shah and Mäntylä 1995)). Fortunately, given the strict geometrical limitations of the pressed PM parts and some carefully chosen simplifications, it can be made feasible, as will be explained below.

### 2.3 The geometry of pressed PM-parts

As already concluded, a pressed PM part needs to have a limited number of levels formed by tooling elements in order to achieve an acceptable and evenly distributed density in the part. This is the foundation of the most important feature, namely what is here called a powder column. A powder column is a prismatic body with uniform section. The axis of the powder column is oriented in the parts pressing direction as seen in Figure 4. A pressed PM part can be seen as one or several such columns stacked or placed side by side. Certainly, PM parts rarely consist of powder columns alone. First, some features are possibly formed by face forms. Second, fillets, rounds, tapered sections, and chamfers also commonly appear in PM-parts. The chamfers, fillets and rounds have to be placed considering the function of the tooling. They must not be placed in outer corners because this leads to that the punches must be manufactured with protruding edges (such as shown in the Figure 5), giving unsuitable and fragile punch geometry.

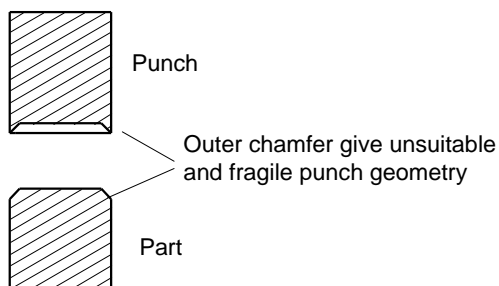


Figure 3. Unsuitable placement of chamfers and fillets

The same problem does not exist in inner corners. If chamfers or fillets are needed on outer corners they are preferably formed in the die as shown in Figure 4 (which also shows the other discussed features).

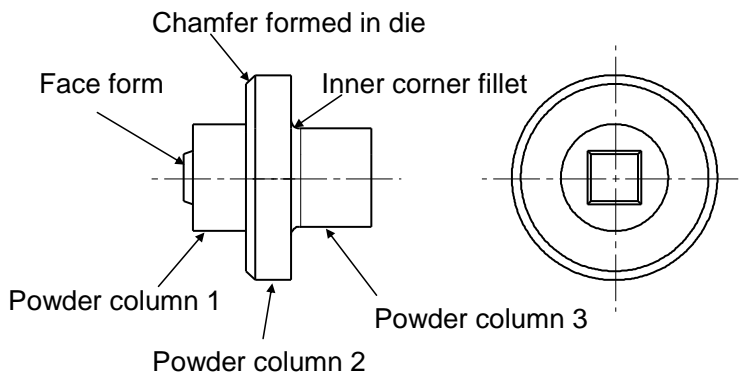


Figure 4. Examples of common features found in pressed PM parts

The process also allows the powder column to be manufactured with tapers, although this also introduces a disturbance in the density distribution. To prevent the tooling elements from jamming they must have a straight part section at one end. Even spherical shapes can be compacted.

Is possible to describe a large number of mainstream pressed PM parts with the above features. It must be clarified however that the above features cannot describe all parts. There are exceptions, often figured out by clever and innovative PM-parts manufacturers. The motivation for the system is, however, to provide an insight for the engineers working on the development of PM-parts. Therefore, it is assumed that it is sufficient to handle the basic rules and recommendations as a first step. To program an experimental system to test the feasibility of the idea in a reasonable time, it is assumed that the PM-parts are created from only powder column features. All other face-form, fillets, tapers, round and chamfer features are for the time being being disregarded.

## 2.4 Reconstructing the parts geometry

In the work of Dissinger, described in Section 1.1, a PM-specialized CAD system tailored for the process was described. The system allows the designer to sketch the profiles of the powder columns interactively in a specialized sketch program. The sketched profiles were then analyzed for their manufacturability and subsequently extruded. If instead any 3D CAD-model suggestion is going to be evaluated automatically, the same principal can be used. Rather than letting the user sketch the profiles, they can be directly identified in the model. This is done by searching the CAD-model data-base for edges that form closed loops at constant or nearly constant levels, i.e. planes, perpendicular to the pressing direction. When these closed loops have been identified they can be used to reconstruct the model in the receiving CAD-system in two steps. It begins by fitting geometrical elements such as parametric lines and arcs to the in-plane edges of the CAD-model. With the parameterized sketch in place and knowing what side of the loop there is material in the neutral solid model, a parameterized version of it can be reconstructed.

### 2.4.1 Creating parameterized sketches

The fitting of geometrical entities to the model edges is done provided that the whole edge is located in the x, y plane. In other words, all points located on the edge must have roughly the same z-coordinate meaning that a level has been found at that location. The fitted geometrical entities are thereafter grouped to form connected and closed loops. To form the CAD-model there must be one or several such loops in each level, all connected and closed. Figure 3 below shows a sample part where eight different loops have been identified on five different levels.



For example, on the level  $z=-8$  there is one loop present containing line1, arc1, line2 and arc2 where A denotes arc and L line.

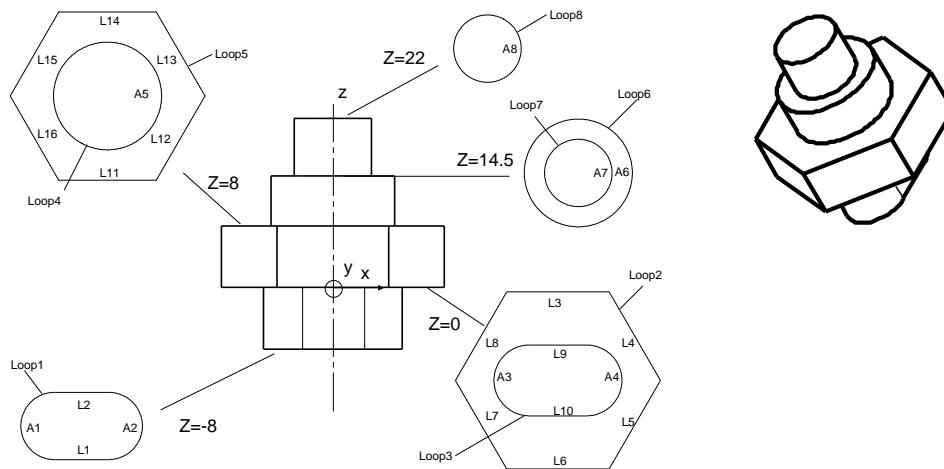


Figure 5. Loops found in example model

The table on the next page shows the loops and what levels in the global model coordinate system they belong to.

Level	Loop
$Z=-8$	Loop1
$z=0$	Loop2, Loop3
$Z=8$	Loop4, Loop5
$z=14.5$	Loop6, Loop7
$Z=22$	Loop8

Table1. Loops found in each level

Similarly, the line and arc entities belonging to each loop and the points belonging to each entity are also kept track of.

#### 2.4.2 Reconstructing the Solid model

Having extracted all wire loops from the neutral file, the solid geometry of the part can be reconstructed if it is known what side of the loop there is material in the original solid model. This information is appended to the loops. If two or more loops appear on the same level (as is the case on sample model levels  $z=0$ ,  $z=8$  and  $z=14.5$  in the Figure 3), two different cases, as shown in Figure 6, can occur. Either one of the loops is completely contained in the other (case 1) or the loops are completely separated (case 2). For topology reasons the third case, where the loops are partly overlapping, cannot occur. It was a valid solid model in the first place, and letting the loops overlap would produce an invalid model. If case 2 is found, the material side is determined by examining any point located between the two loops.

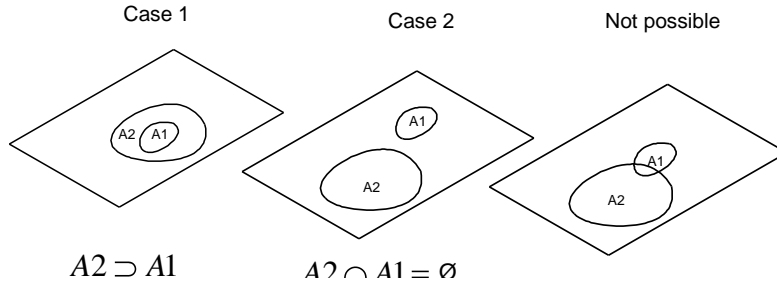


Figure 6. Loop arrangements in planes.

For the reconstruction of the solid body, all loops are linearly extruded to the full height of the part and then material is removed in the opposite direction of the loop or pair of loops material directions as shown in Figure 7. Now a parameterized model have been built in the target CAD-system allowing the rule-base to easily retrieve and evaluate the parameterized geometric elements so that a number of geometrical conditions important for the parts manufacturability can be checked.

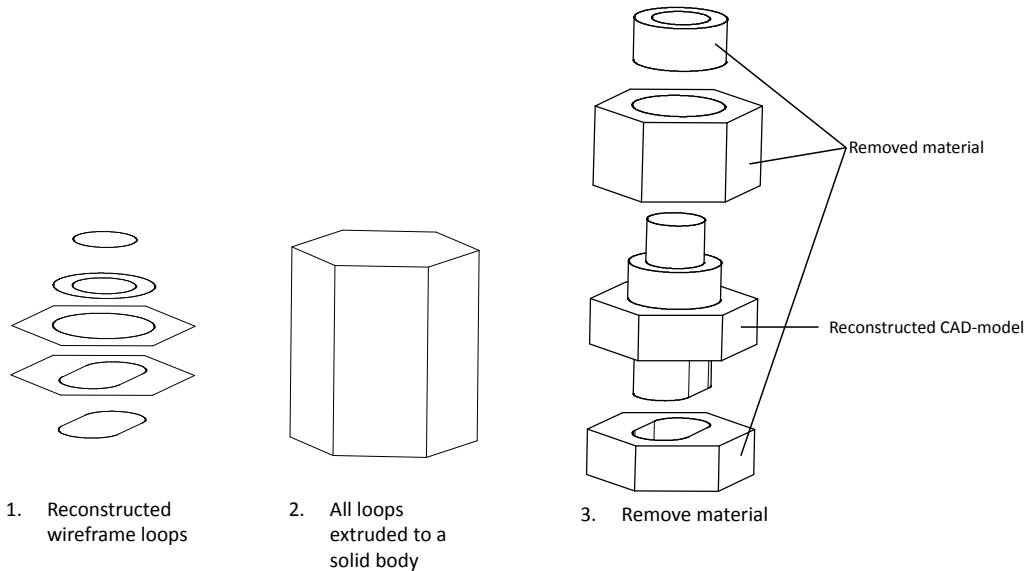


Figure 7. Reconstructing the sample model from Figure 3 in the receiving CAD-system.

### 2.4.3 Evaluating the model

The rule-base operates on geometrical data retrieved from the reconstructed CAD-model. To show this possibility, a number of the general rules and recommendations found in the handbooks earlier referenced have been represented. It is, for example, checked that:

- The individual loops do not contain narrow sections to prevent fragile tooling elements and impaired powder filling.
- The angles between all geometrical entities in the loops are sufficiently large. A small angle indicates the presence of feathered edges.
- The distance between the individual loops in each level is sufficiently large, so that tooling elements that are too thin-walled will have to be used.

- The diameter/height ratio for each powder column does not exceed the minimum permissible value causing unacceptably low density near the neutral layer.

The format of the rules is: If <condition> then <reaction>. An example is then determining the presence of feathered edges. First, all angles between the lines and arcs in the sketches are measured in the reconstructed solid-model. These are readily available in the construction history tree of the parameterized model. If any of the angles should fall below a threshold value stated in the rule, the user is alerted on this condition. The threshold value can be accessed by the user so that it can be adjusted to the conditions of the specific type of process and product the designer is working with. These and other geometrical checks are made. If any warnings are received, the model has to be revised and re-evaluated until it passes all checks.

#### 2.4.4 Example parts

In order to verify that the system accurately can reconstruct the CAD-models, a number of PM parts have been tested. Some of the parts have been obtained from the Höganäs handbook (Höganäs 2007) Chapter 8. and others from the production of PM parts at the Swedish PM-parts producer Callo AB. The parts are shown in Table 2 on the next page.

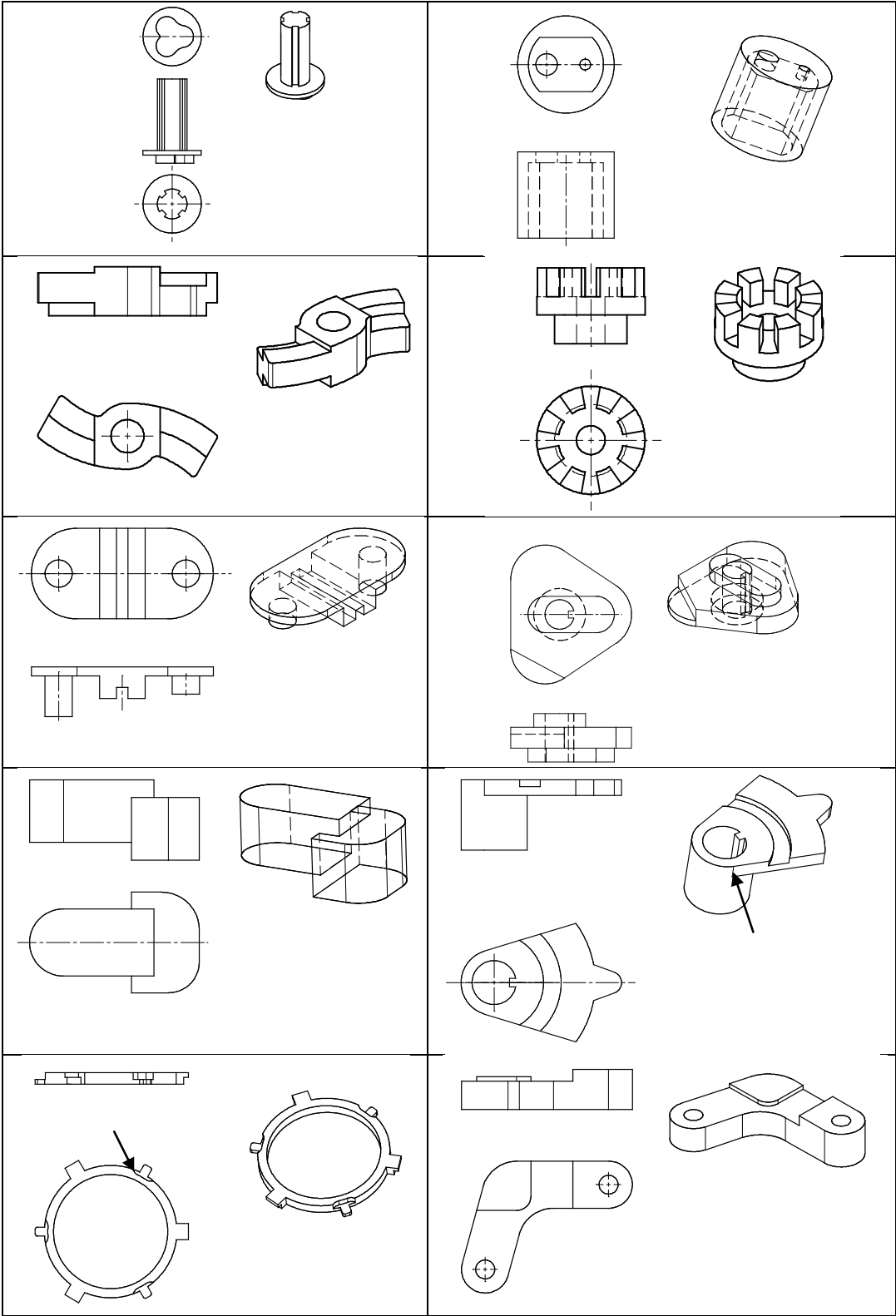


Table 2. Parts tested

The process had to start with the removal of all secondary features, leaving only the powder columns. All the above parts passed the rule-based checks except the ones with arrows indicating that too small angles were found indicating the presence of a feathered edge.

In the receiving CAD-system, the designer can check that the reconstructed geometry is identical to the original. Should there be any differences these can be corrected directly in the reconstructed CAD-model prior to starting the rule-based evaluation.

## **2.5 Automated tooling design**

So far, only general geometric recommendations have been addressed. If any more detailed evaluation of a proposed geometry is to be obtained, the planned tooling setup must be known. This cannot be derived directly from the geometry. The planning of the tooling assembly design requires many other considerations such as:

- Should a certain feature be formed by a separate die or is it possible to use a face-form?
- Should a stepped die or separate punch be used?
- Are the lengths of the planned tooling elements sufficient for the whole pressing cycle, filling, pressing and ejection?
- How should the tooling elements move during the cycle?
- How should the part be oriented in the press, and will there be any powder transfers needed?
- Can the tooling elements withstand the mechanical loading?
- Can the needed tooling motions and forces be obtained by the press?
- Is the tooling assembly subjected to unsymmetrical loads while pressing due to asymmetry of the part?

Clearly, the automated tooling design process has to involve the requirements of the part. It should however be possible to generate a few alternative tooling configurations to provide a basic understanding. This will also enable an indication of the expected properties to the designer so that the relationship between the requirements on the part and the tooling setup becomes clear. One example of a consideration is that a feature for which good mechanical properties are needed should not be formed by a face form unless secondary operations such as re-pressing or coining are planned. Understanding the alternative tooling configurations will perhaps mean that the parts can be redesigned with a more well-planned leveling, getting the most appropriate tooling design for the part. The designer will get a set of geometrical recommendations for the design suggestion and also one or more possible tooling setups. If the part has been designed with an unnecessary large number of levels, seeing the corresponding tooling design will alert the designer as to why the design is unreasonable and needs to be revised.

In order to make this preliminary planning of the tooling, the reconstructed CAD-model is thought to be useful. Since a number of parameterized sketched loops have been derived and it is known at what side of these loops the material of the parts is, then in the opposite direction there must clearly be a tooling element. The length of this tooling element cannot,

however, be determined without knowing the powder, the planned adaptor device and the exact production cycle. However, since the goal is just to make the designer aware of what impact a certain design will have on the tooling setup generating a model or picture, showing the setup is assumed to be sufficient.

In the sample part shown in Figure 8 below, the identified closed loops have been directly extruded to form the tooling elements. Establishing the detailed tooling design, however, as shown on the right side of the figure, had to be done manually.

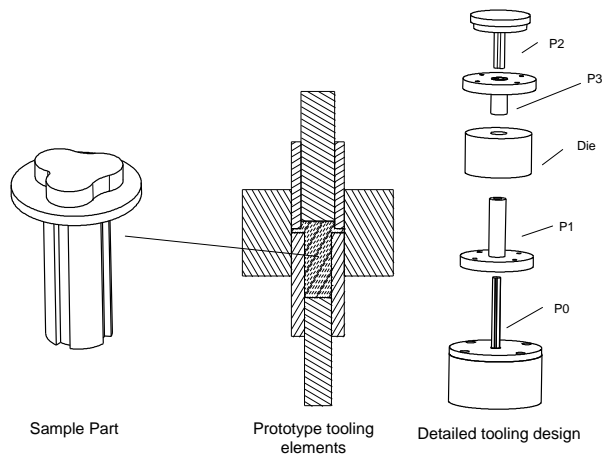


Figure 8. The closed loops and their material directions used for the creation of a preliminary tooling assembly.

### 3. CONCLUSIONS

This paper has shown how to use the functions provided in a commercial CAD-system to automatically evaluate proposed PM-parts designs for their manufacturability. A first step towards the automated tooling design to visualize the alternative tooling designs to the designer has also been taken. These types of systems have, as mentioned, previously been described in literature. However, the automated construction of history tree and parameters directly from any CAD-model in such way that its geometrical elements and their parameters can be made available is, to the author's best of knowledge, new. Other important conclusions from the above study are:

- The ideal pressed PM-part can be described using a limited number of standardized features based on the restrictions of the pressing process. The features have direct counterparts in a commercial CAD-system.
- The primary features, the powder columns, can be automatically identified by searching the model for connected loops of edges in CAD-model. These can automatically be reconstructed in the receiving CAD-system allowing an open and user revisable rule-base to evaluate the manufacturability of them. The rule-base can be revised by the user to reflect the build-up of new knowledge.
- The function of the tooling assembly needs to be known to make any prediction regarding the expected density and its distribution. Therefore, automated design require knowledge about the requirements of the envisioned part.

- When finding alternative options for the tooling assembly design, the automated retrieval of the levels is expected to find a starting point for the tooling solution.
- Many in-production PM- parts can be rapidly evaluated using the proposed methodology. By comparing the results of the evaluation with the actual outcome, a refinement of the geometrical recommendations to include the process, product and the material can be made.

#### 4. FUTURE WORK

In the study, the parts have been idealized, disregarding secondary features and only addressing the powder columns. In order to provide more detailed advice, the effect of these must also be incorporated. This requires that the system be extended to identify and reconstruct them. This is planned to be done in steps, starting with adding the features that are most frequently found in the PM-parts. This requires that an increased number of PM-parts cases be studied.

In tooling design, in most cases, there are several alternative tooling setups for a given geometry, resulting in different part costs and properties. It is envisioned that this has to be based on previously completed tooling design cases, connecting the reconstructed CAD-model to a particular case possibly with additional input from the designer. Further, having a working version of the system and presenting it to real life designers in industry is expected to provide new insight into exactly what type of support it should be focused on.

#### 5. ACKNOWLEDGEMENT

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The author also wishes to thank Professors Staffan Sunnersjö at the University of Jönköping and Lars Nyborg at Chalmers University of technology for initiating and leading the research project "Regelbaserat system for sintergodstillverkning (In Swedish)" financed by the participating companies and research grant "Effective product realization" sponsored by the Swedish governmental agency for innovation systems, Vinnova.

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# MANAGING PRODUCT AND PROCESS KNOWLEDGE IN CAD-SYSTEMS

ROLAND STOLT

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## ABSTRACT

In order to codify, manage, and reuse the knowledge gained from pursuing product development projects, CAD-programs have started to provide functionality for supporting this activity. Since the primary purpose of the CAD program is to interactively define geometry, the functionality has been integrated primarily to be used on the parameters of the CAD-model of the design instance. This paper explores how the functionality can be used in a broader scope, managing and automatically reusing knowledge for all designs intended for a certain production process. This is facilitated by automatically converting an arbitrary CAD-model of a design to a format that is accessible for knowledge handling and reuse functionality in the CAD-system. A conversion algorithm for the processes die-casting and powder metallurgy pressing sintering is presented, as well as showing how the knowledge can be represented in the CAD-system. This will allow engineers to actively participate in the knowledge build-up and insure that the general knowledge can be handled in the companies' IT-infrastructure.

Keywords: Production process, Knowledge-based engineering, CAD, Application programming, Knowledge management

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## 1. INTRODUCTION

In any product design process, the knowledge and experience from the development of similar products is highly valuable. Consequently, there is great interest in gathering this knowledge and structuring it so that it can be used in later product development projects. This is known as knowledge management (KM). The traditional view of KM is that knowledge exists as an asset within the organization. A knowledge engineer can be sent out to interview domain experts representing different areas of expertise. Examples of such areas include mechanical design, electrical design, and production. The role of the knowledge engineer is then to formalize the knowledge so it can be codified by a programmer for reuse. Lately, this view has been modified somewhat (Swan, Newell et al. 1999) to acknowledge that knowledge is constantly being developed and is not just something static that can be gathered and implemented in computer code. One must also take people and processes into consideration. This is described by (McMahon and Browne 1998) as the codification versus the personalization strategy of KM.

Depending on the view of knowledge, the IT-tools for supporting it vary. Some emphasize the collaborative work (such as work-flow in PDM systems). Others stress the codification and later reuse of the knowledge. In knowledge-based engineering (KBE) systems,

codification is emphasized. KBE provides good support for codifying and reusing knowledge. However, it is challenging for the engineers to structure the engineering knowledge gained from pursuing product development in collaboration and codify it into the system. This has, as mentioned, traditionally been the role of knowledge engineers and programmers. However, if the role of the engineer changed to participating in the gathering and codification of the product and process knowledge, he or she would take an active part in the KM so that the codification and personalization strategies could be joined. This way of working is supported by CAD-system developers who have recently started to provide KBE functionality integrated in the CAD programs. This allows the engineers to work in a familiar environment (in other words, the CAD programs allowing them to include knowledge in their CAD-models). Since the CAD program is often a central and integral part of the companies' IT infrastructure, this will provide a way of conducting KM naturally integrated with other IT-tools. However, the basic and traditional function of the CAD system is to define the geometry of the product interactively. Consequently the KBE functionality has been integrated bearing in mind that the resulting software is to be used for working interactively with the geometry definition. If the CAD-system is going to be used generally for managing knowledge (not only for the specific instance currently being designed, but also to be applied with a degree of automation on similar products sharing the same functionality and production method), an additional interface is needed. This interface will connect the design instance with the general product and process knowledge, allowing the functionality of the CAD-system to be automated and applied in a more general scope. The CAD-system will then be set to act as a KM-tool.

The interface, suggested here, is based on the fact that general knowledge exists about the manufacturing process. This knowledge is not likely to change since it would alter the general concept of the production process. Since a parameterized CAD model is needed for accessing the KBE functionality of the CAD-system, the general production process knowledge can be used to derive a construction history and parameters just as if the geometry had been interactively defined. Specialized knowledge can be applied to this reconstructed model through the integrated CAD/KBE system. The process, as depicted in Figure 1 below can be applied to the majority of parts sharing the same production process.

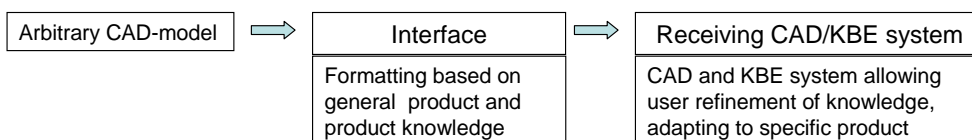


Figure 1. Interface between CAD-model and CAD/KBE system.

The reconstructed CAD-model is an alternative representation of the geometry with a known structure, such that a CAD/KBE system can be used for the purpose of generative design or design evaluation, for example. The automated construction of CAD-models with specialized parameters and construction histories in order to access the knowledge management (KM) and reuse capabilities of the CAD-systems have been described by the author of this paper earlier (Paper IV). It then involved powder metallurgy (PM) pressed and sintered parts. Also die-cast parts have also been explored (Paper II). However, it was based on that the features of the part be constructed interactively according to a modelling instruction. This lead to a CAD-model with a parameter and feature structure prepared for subsequent automated design tasks. The design task involved the construction of target surfaces for shell element meshing. It was discovered while developing the systems that it was difficult to model complex die-cast geometries using only simple commands. While working with PM parts a simple feature recognition (FR) method was developed. That meant that the specialised

CAD-model automatically could be created from any CAD-model. The purpose of this paper is exploring how the FR method applies in the die-casting case and also to show how the CAD-systems can be used for KM and reuse both in the die-casting and PM case. It is first necessary to provide the reader with an insight in how FR method works in the PM case. This was introduced in paper IV, and will in this paper be extended to the die-casting case. This is done in Chapter 2. The KM and reuse in the CAD/KBE systems is described in Chapter 3. Finally, the conclusions are presented in Chapter 4, and Chapter 5 suggests future work.

## 2. FEATURES AND PRODUCTION PROCESSES

In the context of this work, it is assumed that an arbitrary format CAD-model describing a design suggestion exists. It may originate from an external source or in-house. The role of the system is then to automatically evaluate the geometry from some aspect such as its manufacturability. It may also be to further develop the geometry through automated generative design. A typical example is when a customer has sent a design suggestion for quotation or to get feedback on its manufacturability. Now, in order to apply the product and production process knowledge on the design, an interface to the CAD/KBE system is, as mentioned, needed. The proposed geometry has to be given an engineering meaning so that the shape, tolerances and other part requirements can be assessed. Feature recognition been used to realize such interface.

Feature recognition has mostly been used for planning machining operations (Han, Pratt et al. 2000). Lately, feature recognition for the evaluation of manufacturability of parts to be produced by other production methods has also been suggested in research (Lockett and Guenov 2005).

Several methods for feature recognition exist (Shah and Mäntylä 1995). The model, possibly in combination with alternative representations of it, is searched by rules, hints or other methods. This paper does not provide any details. However, regardless of method, the feature recognizer has to be programmed with criteria to unambiguously sort out the sought feature from the rest of the geometry. In order to use the shape restrictions of the production processes, the key idea is to decompose the part into primary and secondary features based on the production process.

### 2.1 Feature identification and model reconstruction

The die-casting process and the PM pressing and sintering processes both have restrictions on what shapes can be manufactured. This made it possible to sort out a limited number of producible shapes i.e. features. Using these features, it was possible to reconstruct the majority of parts produced by the processes. However, the basic concept of the processes can vary by using innovative technology on the production equipment. This is the reason why it is impossible to decompose all parts produced by the process in the suggested way.

#### 2.1.1 PM pressing and sintering

When producing parts using the PM pressing and sintering process, the part must be uniformly compacted so that the resulting density (and thereby the resulting mechanical properties) will become reasonably equal in the whole part. To realize this, it is necessary to design the part with a limited number of levels, each pressed by a tooling element. Consequently, the ideal PM-part consists of a number of columns stacked or placed side by side. Apart from the powder columns, a number of additional features such as fillets, rounds, and tapers can also be manufactured. The process also allows that the front ends of the

punches be shaped to locally form the powder at the ends of the powder column. The height of the features formed this way has to be limited compared to the total height of the column.

Due to the ideal shape requirement, the powder column is the most important feature of the PM pressing and sintering process. The other features can be seen as local modifications of it and can be added if they do not compromise the density and its distribution too much. It is this division into primary and secondary features that is utilized for reconstructing the part in the CAD/KBE system.

### 2.1.2 Die casting

It is also possible to categorize features manufactured by Die-casting into primary and secondary even if it is not as clear as for the PM parts. The ready die-cast part has to be ejected from the die, which consists of two halves, one stationary and one moving. These are called cover and ejector dies and are held in place by platens. Consequently, the parts have to be manufactured with a clearly defined tooling opening direction. Further, the thickness of the walls has to be kept small compared to the total surface area of the part (Andresen 2005). This is in order to give short cycle times.

Should there be a gap between the tooling halves when the tool is closed, a transverse “floor” is formed in the gap. The zone where the tooling halves meet is referred to as the parting line. The orientation of the parting line floor can be allowed to vary as long as it does not block the tooling opening direction. These restrictions lead to that the part can be seen as a structure of features formed in the tooling halves running in the tooling opening direction, possibly disrupted by material formed in the parting-line gap. The picture below shows an imaginary die-cast part describing this distinction into parting-line gap features and features formed in the tooling halves.

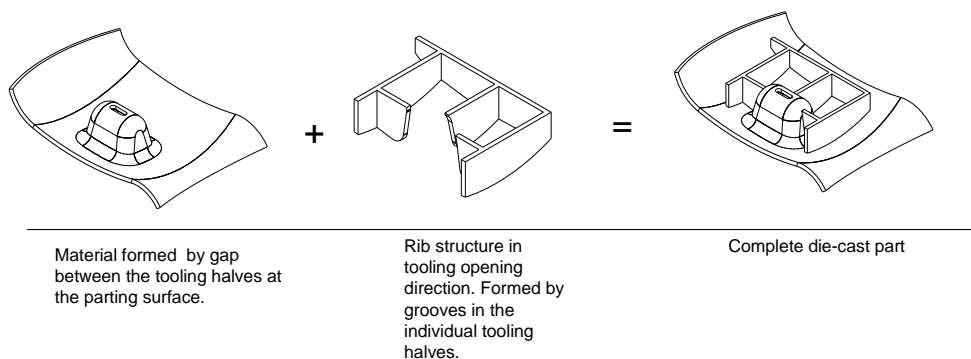


Figure 2. Die-cast part.

As a consequence of the above, the reinforcement rib can be seen as the primary feature of the die-casting process in the context of this work. The parting line floor is an equally important feature. However, since the rib structure is easily identified due to its singular direction, it is especially important in the work presented here. It will therefore be regarded as the primary feature of the die-casting process. It should be noted that the described parts, which may be called ribbed shell structures, is only a subset of all parts that can be manufactured by the process. Solid geometry parts can also be manufactured, provided that the material thickness is limited. Also, the material thickness can be allowed to deviate from the overall material thickness in the ribbed shell structures. Hence, features with an arbitrary section can be formed, provided that the solid section is not too thick compared to the overall wall-thickness. Since the section is arbitrary, it is not possible to predict a priori what types of shapes that will be needed in the particular application. Consequently, the users must be able

to specify their own feature sections. Regardless of the section, the feature has to orientate in the tooling opening direction. Here, this type of feature is here called user defined.

Figure 3 below shows two sample parts produced by each of the processes and the features that the parts can be decomposed into.

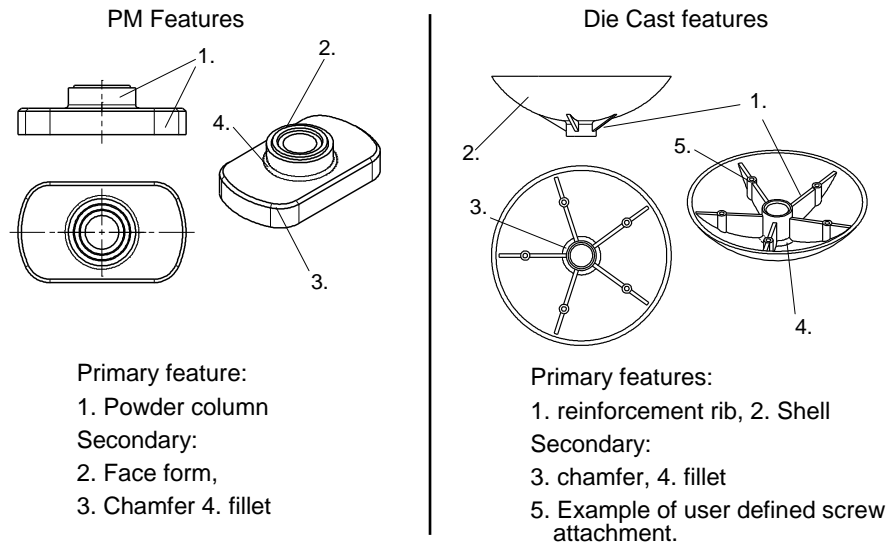


Figure 3. PM and die-cast features for identification

## 2.2 Model Re-construction

The model reconstruction procedure consists of several steps. First, parameterized sketches are constructed on the edges of the models. CAD-models for different purposes can be derived from these sketches. The use of the reconstructed CAD-models will be demonstrated by making a rule-based manufacturability evaluation of the PM parts and an automated construction of a target surface for FEA meshing for the die-cast part.

### 2.2.1 Constructing parametric sketches

The production processes imply that the geometry must be oriented in the tooling pressing direction for the PM parts and in the tooling opening direction of the die-cast parts. The PM parts also display the characteristic levelling allowing decomposition into powder columns. These columns start and terminate on planar or nearly planar faces. To identify the column features in the PM parts, the planar levels are retrieved by searching for edges in the neutral CAD-models that stay in single planes perpendicular to the pressing direction. When found, parameterized geometrical 2D elements (lines and arcs) are fitted to the edges provided that they form closed loops. The reconstructed loops of the PM sample part are shown to the right hand side in Figure 4 (below), which also shows a sample of a die-cast part on the left.

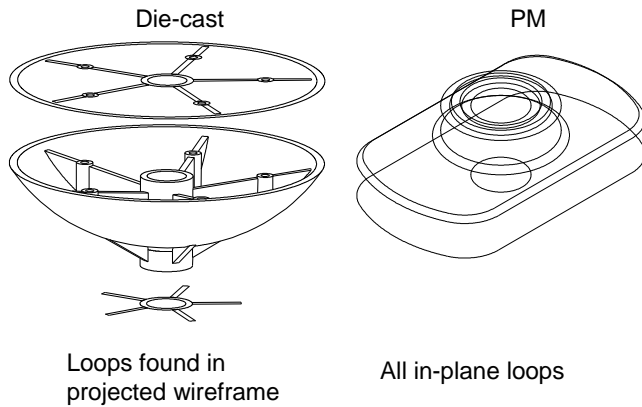


Figure 4. Parameterized loops for die-cast part (left) and PM-part (right).

The loops form parameterized sketches in the CAD-system from which the original solid model can be reconstructed in the PM case. First, the parameterized loops originating from the secondary features (such as chamfers, fillets and face-forms) need to be identified so that they can be isolated from the powder columns, providing a division into primary and secondary features.

The Die-cast parts require a different procedure since they do not have the distinctive levelling of the PM parts. Whereas the powder columns extend between planar faces, the rib features in the die-cast parts extend from their starting surface, which can have any shape up to the parting line or to the end of the part in the tooling opening direction. Discontinuing the feature would mean the presence of an undercut, which requires a side action in the tooling. Since the rib features do not start on planar levels, the edges have to be projected onto planes perpendicular to the tooling opening direction (as shown to the left in Figure 4). Since rib features can be present on both sides of the shell feature, two projection planes are needed, and they have to be located on each side of the part. From these projections the rib and user defined features can be retrieved by comparing the relationships between the individual elements in the projected wireframe. The rib features are identified by finding geometrical items of the same type (lines that run in parallel for a longer distance than twice the material thickness, for example). The user-defined features are identified by characteristic dimensions and patterns of the sketch segments (see Figure 5 on the next page). The retrieval of the die-cast part shell feature has been found to be difficult. It is often divided into several patches by the draft direction features so that automated reconstruction of its mid-surface becomes very difficult or impossible. Further, the shell feature is often double curved, and the inside does not have to have exactly the same shape as the outside. It has been found that it simply has to be reconstructed by the user and tagged in the construction history tree so that it can be identified (Paper II). It should be added that in practice it is often available as a “styling surface” defining the outer appearance of the product. This can, from the designer’s point of view, be seen as a requirement. The designer would then be obliged to design the part to comply with an outer appearance readily defined from start.

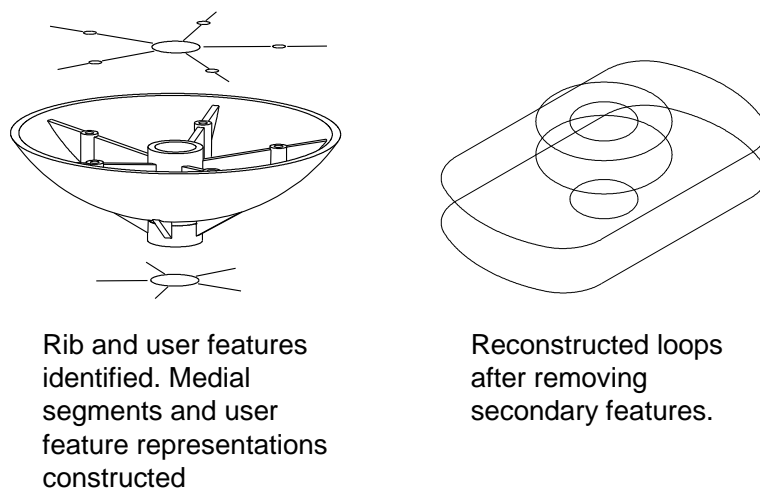


Figure 5. Identifying features in planar projection planes.

Similar as for the PM part, a parameterized version of the feature sketch is reconstructed in the target model in the receiving CAD-system. Since the motivation behind the die-cast system is not to reconstruct the solid model but to make a target surface for shell element meshing, a different reconstruction procedure is adopted. Rather than reconstructing the feature sections, the mid-segments of each feature are derived and constructed in the model (as shown on the left side of Figure 5). The user features (5 in Figure 3) are represented by segments defined by the user. These segments, when extruded to surfaces and meshed with shell elements, will approximate the behaviour of the solid features in the subsequent simulation.

All medial and user-defined segments are subsequently extruded and intersected with the original solid to form surfaces that can be meshed with shell elements

### 2.2.2 Constructing PM CAD-models from the parameterized sketches

For the PM parts, the motivation is the reconstruction of an idealized model for manufacturability evaluation. To reconstruct the PM part, all loops are extruded through the entire height of the part, as shown to the left of Figure 6 below.

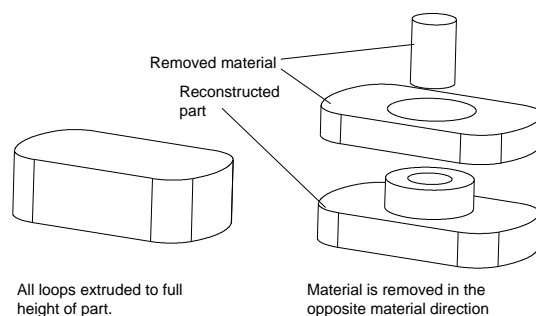


Figure 6. Reconstruction of PM solid-model.

The intermediate loops are then used to remove material. Since it is known which side of the loop the material is located on, the solid is reconstructed by removing material in the opposite direction. A parameterized and idealized version of the part has now been reconstructed. Its parameters can be made available to the integrated rule-base system in the CAD-program. There are geometrical recommendations with the purpose of securing that a functional



tooling can be produced. These are given as geometrical conditions to be checked. A rule-base can be defined that checks the parameterized model so that e.g. the column height is not too large compared to the diameter or the density will be too low in the neutral layer of the powder column. It can also insure that there are no long protruding sections that may break when the part is ejected from the tooling.

### 2.2.3 Constructing a target surface for shell element meshing

The die-cast part mid-segments are extruded as in shown in the centre of Figure 7.

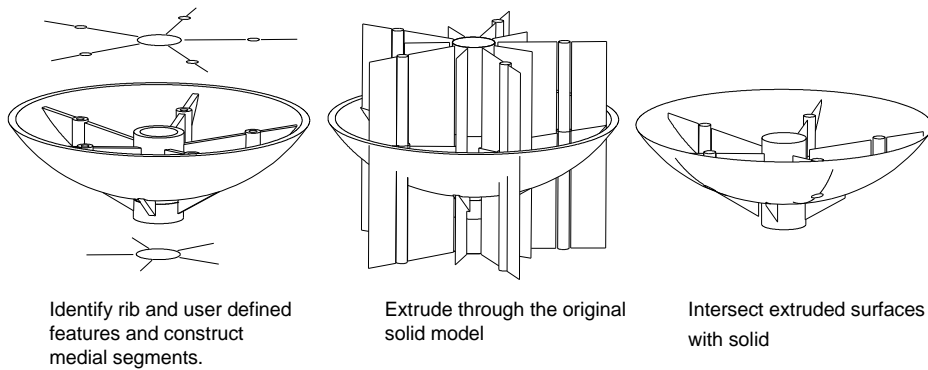


Figure 7. The construction of target mid-surface for shell element meshing

Since rib and user features are present on both sides of the solid-model, the extrusions have to be made from both sides extending up to the shell feature surface. The extruded surfaces are thereafter intersected with the original solid leaving the portion of the extruded surface that is common between the solid and the extruded surfaces. When doing so, gaps or excess surface patches will remain in the joints between the surfaces. In order to have a connected surface, the patches need to be trimmed together. This has been found difficult to achieve automatically, and interactive trimming is often needed. The constructed target surface can now be used for the creation of a shell element FEA model.

## 3. USING THE CAD/KBE SYSTEM FOR KM

This chapter elaborates the possibilities of using the CAD/KBE system for KM. As often is the case in engineering tasks, there are several types of knowledge and representations of it that are used. For instance, all knowledge cannot practically be represented as rules. There is clearly a need for alternative representation forms. In (Cederfeldt 2007) an attempt to map different engineering tasks to an applicable solution technology is made.

The representation of the knowledge in the construction history tree will allow active user-centred management of the knowledge provided that it is extended beyond the instance of the design at hand and it allows different knowledge representation forms to be managed in the same system.

From developing the prototype systems, the items listed below have emerged as being important for the users to control in the studied cases.

- Thresholds and adjusting parameters for the feature recognition and model reconstruction processes.
- Adding, deleting or modifying manufacturability checks.
- Adding, deleting or revising user features, and face-forms.

- Sequentially programmed routines

This knowledge has to be explicitly readable and revisable to the user. Consequently, it cannot be hard-coded into the system. Instead, the CAD/KBE system can be used to manage the above knowledge aspects by using a template CAD-file. The file contains a knowledge-base with the geometry rules, prototype features and a number of parameters for the adjustment of the feature identification and the model reconstruction processes. The sequential code needed for creating and inserting the surface representations in the construction history tree and the reconstruction of the PM-part solid geometry could also be represented as scripts and macros in the construction history tree. However, it would take programming skills to understand and revise them while other knowledge items are more accessible to users only familiar with the CAD-program. Therefore, although displayed in Figure 8, the programs are in this work regular scripts or stand-alone programs. The figure shows the intended structure. Since the knowledge resides in the template CAD-file it can be managed in the regular IT-infrastructure of the company. The version of the CAD-template file can be linked to the correct issue of the CAD-model. It will have the added advantage that the “Knowledge-file” is separated from the parameterized CAD-model. This will allow the model and its parameterization to be sent out externally without disclosing any of the knowledge the geometry was based on.

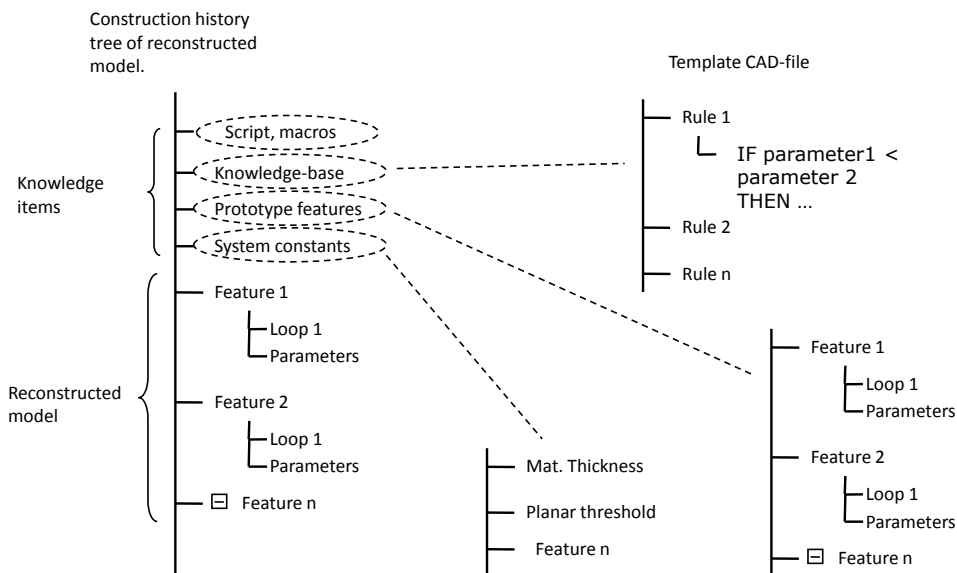


Figure 8. Knowledge features from template CAD-file amended to reconstructed model.

### 3.1 Managing the user features

Every type of product has its own typical shapes specific for the manufacturing process and the conventions developed by the team of designers. There may be similarities in the attachment points for screws between rib features and so on. Their exact appearance, however, cannot be determined a priori. Naturally they have to follow the restrictions of the production processes. The intent is to allow the designer to define the features directly in the CAD-program. Thus, a customizable feature recognizer is needed.

There are examples in literature of customizable feature recognizers such as Custom-Cut (Gaines and Hayes 1999) for machining operations and for general use in CAD systems (Salomons, van Slooten et al. 1994). Apart from these articles and there does not seem to be any customizable feature recognizers, at least not commercially available. Fortunately, considering the specialized demands that exist in the particular production processes dealt

with here, it is sufficient to recognize the feature from its 2D section. In the CAD knowledge template file, such 2D sections can be defined. An application program will then compare features found in the template file to the 2D geometrical entities found in the model for reconstruction. The sketch segments of the prototype features are then searched by the application program to find what types of geometrical items they contain and what dimensions they have. The features are searched in the same order as they appear in the construction history tree of the template file. As an example, one prototype sketch is shown in the Figure 9 below.

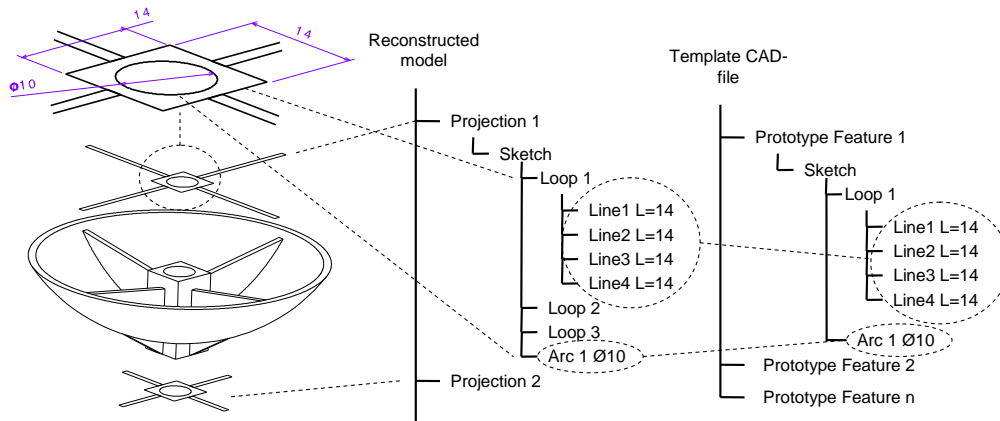


Figure 9. Finding features in projected sketches

It contains one square 14x14 mm and one circle Ø10 concentrically placed. The search algorithm starts by finding elements with the same dimensions (lines of length 14 forming a closed loop of four elements among the projected geometrical elements, for example). The second type of element, the circle, is then sought, and how the loops belong together is subsequently determined by examining the location of their centre. When the 2D sketch geometry of the features has been found, they are removed from the set of projected segments to avoid multiple identifications.

Having identified the feature, different actions can be taken depending on downstream applications. In the example, a target surface for shell element meshing is to be created. An automated design routine is then triggered, replacing the segments with a suitable representation of it to approximate the effect of the solid body entity.

#### 4. CONCLUSIONS

It has been shown how to use the CAD/KBE and KM functionality of the CAD program to manage general product and process knowledge, rather than just dealing with a specific instance of an interactively defined design. This will allow cross project knowledge exchange and user participation in the knowledge build-up and in the codification process. The knowledge template file can be managed in the regular IT-infrastructure. The different types of knowledge needed is represented as rule-based and geometrical knowledge items, will be clearly readable to the user and can be further refined. Further, the proposed way of conducting KM will also lead to the following:

- The reconstructing process can be supervised in the CAD-system, allowing the user to check so that the model was correctly reconstructed and to directly revise it if necessary.

- Rapid evaluation of the in-production products can take place so that refinement of the contained knowledge to comply with already existing products can be made, adapting the knowledge to local circumstances.

## 5. FUTURE WORK

The presented method is centred on the product's geometry. However, there is much other information needed for the evaluation and further design of a product. In addition to the provided geometrical suggestion, a specification of the requirements of the part should also be taken into consideration. This is because it largely effects the decisions made in the product development process. The intention is to continue exploring how the CAD/KBE system can act as a KM-tool for different types of knowledge that extend beyond the specific instance of the design being interactively defined. Further, real practitioners have to be involved. The method requires that the design suggestion be in such state that the feature recognition process can start. Ultimately, if the designer is not capable of creating such a model, the result will be an "unprocessable" message or only a few of the features will be recognized. The question is then how the user can be guided to pass the first step of proposing an identifiable feature structure.

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# APPENDIX

---

```

Function MaterialSide(x As Double, y As Double, z As Double) As String
'Provide a location of the sample point in global coordinates
'Case A Material above loop
'Case B Material below loop
'Case C Both sides
'Case D None
Dim Volume0 As Double ' Original volume
Dim Volume1 As Double 'Volume after inserting a positive direction feature
Dim Volume2 As Double 'Volume after inserting a negative feature
Dim plane1 As Plane
Dim Sketch1 As Sketch
axis1 = CreateAxisData(z) 'A local coordinate systems for the sketch is established on z level
Set plane1 = MakePlane(z) 'A sketch-plane is created on z level
Set Sketch1 = MakeSketch(plane1, x, y, axis1) ' A sketch containing a 1mm circle is created
on the sketch plane with centre point in x,y
Volume0 = MeasureVolume 'Measure the volume of PartBody (Included in appendix)
Call InsertCutFeature(True, Sketch1) ' A positive direction (True) cut feature is inserted
Volume1 = MeasureVolume 'The volume is measured after inserting the feature
Call InsertCutFeature(False, Sketch1) ' A negative direction cut feature is inserted
Volume2 = MeasureVolume 'Measure the volume again
MaterialSide = "D" 'Find the material side from examining the measured volumes
If (Volume0 - Volume1) > 0 Then 'Did the volume of the part decrease?
    MaterialSide = "A"
End If
If (Volume1 - Volume2) > 0 Then
    MaterialSide = "B"
End If
If MaterialSide = "B" Or MaterialSide = "A" Then
    If (Volume2 < Volume1) And (Volume1 < Volume0) Then
        MaterialSide = "C"
    End If
End If
End Function

```

```

Public Function MeasureVolume() As Double
Dim TheSPAWorkbench As SPAWorkbench 'Space analysis workbench
Dim TheMeasurable As Measurable 'The item to be measured must be of type "measurable"
Dim part1 As Part
Dim bodies1 As Bodies
Dim body1 As Body
Dim ref1 As Reference

```

```

Set part1 = CATIA.ActiveDocument.Part

```

```

Set bodies1 = part1.Bodies
Set body1 = bodies1.Item("PartBody")
Set TheSPAWorkbench = CATIA.ActiveDocument.GetWorkbench("SPAWorkbench")
Set ref1 = part1.CreateReferenceFromObject(body1) 'Make a reference of PartBody
Set TheMeasurable = TheSPAWorkbench.GetMeasurable(ref1)
MeasureVolume = TheMeasurable.Volume 'Return the measured volume
End Function

Public Function GetAllEdges()
'The function returns all edges in the model
Dim edges1() As Edge
Dim i As Integer
Dim j As Integer
Dim partDocument1 As PartDocument
Set partDocument1 = CATIA.ActiveDocument
Dim selection1 As Selection
Set selection1 = partDocument1.Selection
selection1.Clear
selection1.Search "Topology.CGMEdge,Body.1" 'All edges in the topology is added to the
selection
j = 0
For i = 1 To selection1.Count
    If Left(selection1.Item(i).value.Name, 15) = "Selection_REdge" Then 'All of type
Rectilinear edge is saved
        ReDim Preserve edges1(j)
        Set edges1(j) = selection1.Item(i).value
        j = j + 1
    End If
Next
selection1.Clear
GetAllEdges = edges1 'Return all edges found
End Function

Public Function Add3PointsOnEdge(edge1)
'The function places a point on the startpoint, midpoint and endpoint of edge1 and returns
the points coordinates.
Dim Points1(2) As HybridShape
Dim HB1 As HybridBody
Dim reference1 As Reference
Dim Hshfac1 As HybridShapeFactory
Dim Coordset1(8) As Double 'x1, y1, z1, x2, y2, z2, x3, y3, z3
Set Hshfac1 = part1.HybridShapeFactory
Set reference1 = edge1 'A reference is created
Set Points1(0) = Hshfac1.AddNewPointOnCurveFromPercent(reference1, 0, True) 'Place
point on startpoint
HB1.AppendHybridShape Points1(0)
Set Points1(1) = Hshfac1.AddNewPointOnCurveFromPercent(reference1, 0.5, True) 'Place
point on midpoint
HB1.AppendHybridShape Points1(1)

```

```

Set Points1(2) = Hshfac1.AddNewPointOnCurveFromPercent(reference1, 1, True) 'Place
point on endpoint
HB1.AppendHybridShape Points1(2)
part1.Update
For k = 0 To 2
    Points1(k).GetCoordinates Coord1 'Read the coordinates of the points to Coord1
    For j = 0 To 2 'To use GetCoordinates the type must be hybridShape
        Coordset1(m) = Coord1(j) 'Coordset1 1 is extended in every loop (counter m)
        m = m + 1
    Next
Next
Add3PointsOnEdge = Coordset1 'The function returns the coordinates of the three points

End Function

```