White Paper

Why Use CATIA V5 for Automotive Tooling?

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Glossary of terms

BKT	Business Process Knowledge Template
BOM	Bill Of Materials
CAD	Computer Aided Design
CAM	Computer Aided Manufacturing
CGR	CATIA Graphical Representation (geometry format)
CNC / NC	Computer Numeric Control / Numeric Control
DFM	Design For Manufacture
DMU	Digital Mock-Up
ERP	Enterprise Resource Planning
FEA	Finite Element Analysis
KBE	Knowledge Based Engineering
MRP	Material Requirement Planning
OEM	Original Equipment Manufacturer (e.g. Car maker)
PDM	Product Data Management
PLM	Product Lifecycle Management
VPM	Virtual Product Manager

Integral Powertrain Ltd

Integral Powertrain Ltd is an automotive powertrain engineering consultancy based in Milton Keynes, United Kingdom. Formed in 1998, it provides a wide range of services, specialising in powertrain engineering and the design of associated tooling. Tier one supplier status is held with a number of OEMs.

Integral Powertrain is a world leader in the application of CATIA V5 to powertrain engineering, having developed advanced processes to rapidly create 3D CAD geometry for major components. Tooling models are produced in parallel with design data and kept up-to date through automatic links, enabling faster tooling design. Tooling is produced using CNC machining methods.

Building on the experience gained in it's own business, **Intrinsys** offers a PLM consultancy service. Advice and training is given on advanced CAD processes, data management, knowledge-based engineering and other associated disciplines, helping companies develop their own skills.

1 Introduction

There can be no doubt that automotive tooling manufacturing is a challenging business. Overcapacity, increasing globalisation and aggressive OEM purchasing strategies are driving intense competition. In order to secure the contracts they need therefore, suppliers must offer advantages to the OEM whilst maintaining the quality levels now required across the industry. These advantages can be related to cost reduction, lead time improvement or a desirable ‘unique selling point’.

In many cases competitive advantage is already flowing from improved communication and working patterns. A more integrated approach often avoids unnecessary duplication and provides improved decision making and work flow when dealing with OEMs, sub-suppliers or even different areas of the business itself.

Another source of competitive advantage is the development and use of advanced engineering and business processes. These can capture and re-use knowledge and expertise and can automate routine tasks leading to substantial time compression and efficiency gains.

In order to achieve these communication and process benefits it is vital that an appropriate CAD solution is in place. CATIA V4 has established itself amongst the OEMs. CATIA V5 now builds on this with user friendly, leading edge capabilities aimed at improving business performance. Furthermore the ability of CATIA V5 to run on Windows™ as well as UNIX operating systems has helped to widen the appeal of the product to smaller companies.

This paper shows how CATIA V5 can be used to help automotive tooling companies win in a competitive market. It starts with a review of current best practice and a discussion of typical areas for future improvements.

The next section considers how to construct a roadmap for identifying and achieving these benefits. This covers technical requirements related to CAD functionality, data management and hardware and software issues. Also, once decisions have been made, the requirements for migration and training are discussed.

The paper concludes with a case study and an accompanying ROI study demonstrating the business case for its adoption.

2 CAD/CAM in tooling design and manufacture

It is worthwhile reviewing how the use of computer aided design and manufacturing (CAD/CAM) has evolved in the tooling industry, and what sort of working relationships exist between component designers and tooling companies.

In the past, component designers worked in 2D and tooling manufacturers interpreted their drawings to manually produce tools. With the arrival of Computer Numeric Control (CNC), machine tools could be driven from 3D surfaces contained in CAD models. Unfortunately, due to software limitations designers were producing models that were insufficiently complete to be cut directly; tooling designers had to painstakingly edit their surfaces to fully define the part (with draft and fillets etc.) for CNC surface machining. In many cases this continues to be the norm, resulting in a serial design process as shown in figure 1, considerable duplication of work, and two sets of component data.

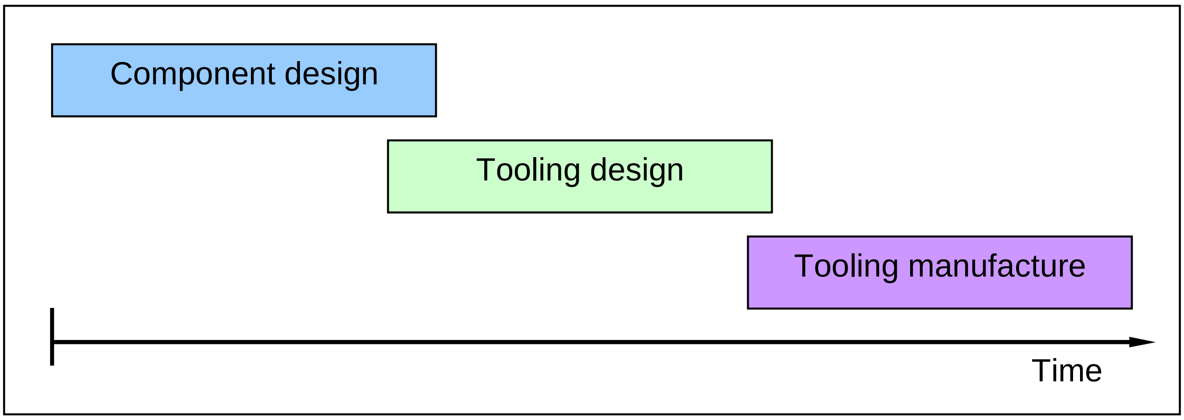

Figure 1 – Traditional tooling design process

There was good reason for tooling designers to use surface-based, "non history supported" CAD systems. History support requires storage of the 'history' of how a piece of geometry was arrived at, which places extra demands on hardware. Working without history meant hardware requirements were less demanding, and it was possible to create almost any geometric form in situations where history supported solid modelling failed. Customers rarely wanted the tooling models and there was no alternative way of working.

Now component designers have CAD systems with history support, proven full definition modelling capability and the ability to share work between multiple users. If a change is made, the modeller simply re-generates the model and the down-stream effects of the change flush through, including draft and fillet specifications. Furthermore, for reasons discussed later, component designers have an interest in owning fully defined geometry which is exactly reproduced in the tool. History supported models have tangible value to the designer.

It makes sense that this geometry should only be created once, in a manner which can quickly be edited to accommodate design changes, and used by all parties concerned. This technology has been embraced by a number of OEMs and their suppliers, who have developed ways of working concurrently on a single data set to dramatically reduce lead-times, reduce the cost of change and eliminate re-work (See figure 2).

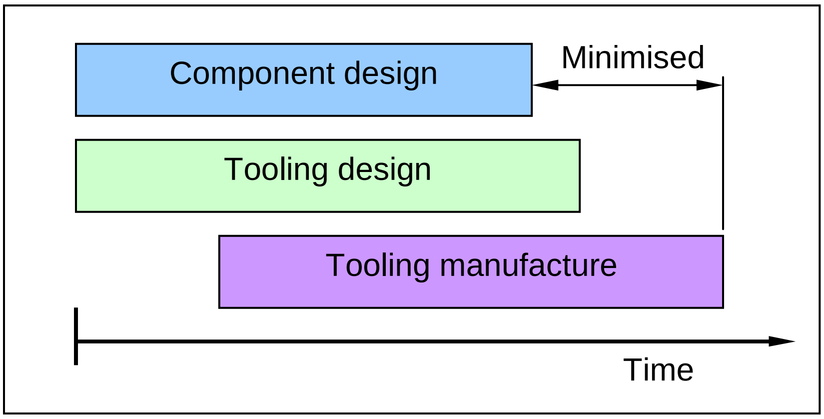

Figure 2 – Parallel tooling design process

Two distinct approaches have developed. The first is for the tooling supplier to build full definition CAD models under the direction of the designer, and develop the tooling at the same time. The second approach is for DFM features to be incorporated by the customer into the component models through close customer-supplier working.

Both approaches require change on the part of the customer and tooling supplier to effectively integrate the skills of both parties, and commitment to a closer working relationship. Furthermore the tooling companies need to invest in systems which can accommodate the continuous change inherent in a concurrent engineering process, and support their customer's requirements.

3 Why adopt CATIA V5? “A vision of the future”

Upgrading a CAD system may provide a number of worthwhile, short-term benefits. These may be related to improved data transfer with a key customer, faster modelling, or other useful features. Since the new system will be expected to last many years however, it is important to have confidence that it will not hold the business back in the future. A company may decide therefore, to identify how it wishes to utilise advanced CAD functionality in the longer term and produce realistic development plans. This section aims to stimulate ideas by discussing the sort of advanced processes that are currently being developed on CATIA V5.

3.1 Parallel design and manufacturing (‘Multi-Modelling’)

The evolution towards sophisticated parallel working practices between customers and suppliers is discussed in the previous section. The available benefits are clear but managing the process can be challenging. CATIA multi-modelling is ideally suited to this type of problem. An advanced, change tolerant structure for parallel working is presented in section 4.2.4 and this has already been proved on complex castings.

3.2 Parametrics

Where topologically similar components are required more consistent designs can be produced in much shorter times using parametrically driven templates. Clearly this reduces the amount of resource spent actually creating geometry and enables models to be driven by those not expert in modelling skills. The potential for parametric templates is partly determined by the CAD system chosen. CATIA V5 has been proven on highly complex parametric models and has advanced functionality to adjust model topology depending on defined circumstances. Its ability to communicate with other Windows based programs also enables user interfaces to be produced quickly and easily.

3.3 Knowledge based engineering

The re-use of knowledge within an organisation is seen by many as a key opportunity for improving efficiency, lead time and the robustness of the engineering process. CATIA V5 has extensive functionality to support this known as ‘knowledge based engineering’ (KBE). These basically enable CATIA V5 to operate as a programming environment. Once geometry has been designed parametrically, it is possible to incorporate design rules to set parameters or govern relationships between them using KBE functions so that much of the design is automatically generated to current best practice.

3.4 Communications

Improved communications may be the key to a number of business improvements. CATIA V5 has great flexibility in this area enabling measurements and calculations to be taken easily and posted into other programs or systems. Some examples are:

3.5 Cost model

Cost drivers such as raw material and machining time are calculated enabling an objective measure of cost to be built up. The model can be used to increase speed and robustness of the bidding process. It can also be used to engage the customer in a rational process to develop low cost solutions (rather than just reducing margins) and deal with variations.

3.6 Resource planning

A similar approach can be taken to resource and facility planning. Parameters that drive these such as design time, machining capacity etc. can be extracted and used to drive company scheduling.

3.7 Outsourcing work

Global sourcing is offering potential to reduce the cost and lead time of machining provided capacity can be located and contracts agreed quickly. This depends on communicating requirements effectively to a number of potential suppliers. Much of this information can be generated via CAD and provided in an 'Outsourcing Pack' enabling suppliers to commit to cost and timing.

3.8 Design validation

Virtual validation techniques are likely to continue to reduce the need for physical validation, or at least to reduce the likelihood of serious concerns developing once hardware has been made. For many businesses this will be a key part of their time compression strategy. CATIA V5 has a comprehensive set of design validation tools including FE analysis, kinematics simulation, clash detection and draft analysis, which can be rapidly and routinely applied using the Master model geometry. In many cases this dispenses with the need for lengthy specialised analysis using dedicated models. Instead analysis can be thoroughly integrated into the design process. Even where specialist analysis codes are required, CATIA's auto-mesher can provide a reliable, rapid and repeatable geometry-based method of data preparation.

3.9 Template models

The ultimate level of automation may be a combination of all of the above capabilities in a template model. The basis of this is a parametric model driven by knowledge-based rules. Many of the optimisation and validation processes are streamlined to minimise the cycle time for optimisation. Communication with other business systems is similarly improved. This concept has great potential in some areas of automotive tooling design and has been successfully applied to automotive crankshafts by Integral Powertrain Ltd.

4 Defining a road-map for change

4.1 Business review and process definition

Once a decision has been made to investigate introducing or upgrading a CAD system, the process for doing this must be established. This must include the building of a detailed knowledge of the business as well as the potential CAD solutions. The opportunity for benefits to extend beyond the design/engineering side of the business should also be investigated before a plan is developed.

Often this plan will itself be the subject of an ROI study before a decision to proceed is taken (see section 5).

The rest of this section describes a proven approach.

4.1.1 Review background and define business objectives

The process starts with a general review of products, markets, competition, sites, number of employees etc. The high-level business objectives driving the need for change are also identified. Typical examples are improved profitability, growth of market share or penetration of new markets. The plan will have to show clearly how it supports these.

4.1.2 Review business process

The existing business process is then reviewed.

It is useful to start by defining the activities throughout the Product Lifecycle, such as:

- Bidding
- Design
- Validation
- Manufacturing
- Commissioning
- Change Control
- Maintenance and Service

Major supporting activities are also considered such as:

- Project Control
- Purchasing
- Resource and Facility Planning
- Control and distribution of data

The activities within these areas and the interfaces between them are then mapped out to build a detailed picture of how the business operates.

4.1.3 Identify improvements

The objective of this activity is to define modifications to the business process that support the business objectives. This activity requires up-to-date knowledge of CAD capability and best practice solutions. It may also be appropriate to make use of benchmark information from competitors or other industries.

As part of this process a number of questions will have to be answered such as:

- Is there a preferred method of working?
- Do customers have a preferred way of working?
- Does the business need to be more closely integrated with it's customers?
- Who needs to access CAD information?
- Who might benefit from being able to access CAD information?
- How is design data validated?
- How is issue control enforced and can this be streamlined?
- How is non-CAD data controlled?
- Where is company expertise or knowledge held?
- How is continuous improvement assured?
- What checks are in place to eliminate duplication of work?
- How are quality procedures assured?

4.1.4 Develop a solution

At this point a detailed solution can be developed for the key areas. Essentially this involves matching functionality to process requirements within a manageable and integrated framework and, importantly, defining how to implement the required changes. A brief description of the key areas for consideration is given below. Sections 4.2 to 4.8 give some more detailed background to each area with respect to CATIA V5.

- **Improved process using CATIA V5 modelling features**

  This will cover key CATIA V5 functionality for creating data and promoting team working

- **Design validation**

  Identification of analysis and digital mock-up requirements

- **Data management**

  How will data be held? Will a PDM solution be required? If so, which one?

- **Hardware / software**

At this stage it is possible to determine the specific hardware and software requirements.

- **Training and outsource support**

Achieving basic CATIA V5 competence is a rapid process. In order to maximise benefits however it is necessary to follow this up with methodology training. In this way best practice can be effectively introduced to the organisation.

- **Continuous improvement**

CATIA V5 will support the evolution of new efficient practices well into the future. In order to take advantage of this there must be a means of directing and driving on-going change.

- **Migration**

Ensuring existing company systems can be operated and that communication routes with customers and suppliers are functioning. Scheduling key decisions and activities so that disruption to business operations is contained.

4.2 Improved process using CATIA V5 modelling features

CATIA V5 has a number of features to enhance the tooling design process. These range from individual functions in the standard workbenches to complete products. If used in conjunction with appropriate methodology, these increase the speed and robustness of the design process. They also support advanced, concurrent working processes and enable rapid and efficient implementation of changes.

The following sections introduce the main CATIA V5 features appropriate for tooling design in the light of their impact on process.

4.2.1 Parametric modelling

One of the most productive features of CATIA V5 is the use of parametric modelling; the ability to drive geometry with dimensions. This means a common model structure can be used as the basis for topologically similar components and promotes speed and consistency. Using appropriate methodology, CATIA V5 has proved capable of producing complex models with exceptional stability over a wide range of dimensions. This parameterisation can be seen as a critical step towards design automation and re-use, which is covered in section 4.7.

The creation of fully parameterised models may in some cases be quite problematic reducing flexibility and tolerance to change. Uniquely amongst the major systems however CATIA V5 enables parametric geometry to be combined with explicit geometry in the same model. This means that parametric elements can be introduced as it becomes practical to do so but without compromising the possibility to customise, tolerate change and maintain timing.

4.2.2 Assembly design

The Assembly Design product is central to all facets of tooling design in CATIA V5. By establishing links between models it enables 'design in context', or the ability to position, drive and assess a model using other models. Clearly tooling and component features can be linked in this way but the usefulness of this feature extends into 'digital mock-up' (DMU) enabling efficient execution of validation activities such as clash detection and kinematics checks. Users also have the ability to manage the assembly structure, generate bills of materials, and access a wide range of design review tools (discussed in section 4.3.1), all from the same workbench.

This capability is available even for assemblies of large complex parts by making use of the 'visualisation' mode. This produces a 'light-weight' (tessellated) representation of the geometry, which places less demand on the computer hardware.

Unlike CATIA V4, Assembly Design is a standard product shipped with most CATIA V5 design configurations. Moving between Assembly Design and Part Design is seamless, with both sharing the same specification tree, facilitating part design in an assembly context.

Much of the functionality discussed in the following sections benefits from being used in an assembly context.

4.2.3 Multi-Modelling

Multi-modelling is the name given to a CATIA concept, which uses the link functionality to enable work to be shared between multiple designers. It can also be used to support the robust and efficient parallel processes referred to in section 2 and section 3.

The basis of multi-modelling is the capability, at all levels of CATIA V5 part design and surfacing, to create links between models. When links are created one model is a parent and the other a child. The 'parent' item contains all the history required to define its topology. The child contains an identical topological representation, but in place of the history there is a link reference containing details of the parent. When the parent changes, the child's reference goes out-of-date and is automatically highlighted as in need of synchronisation.

Removing the history in the child model reduces the data-size. Ultimately it enables a complex model to be created (containing many linked entities) which is both data-size efficient, and history supported.

The following sections illustrate how multi-modelling can be used to facilitate parallel working on a complex piece of form tooling such as a cylinder head casting tool set.

4.2.4 Multiple designers working on the same component

For large and complex components substantial lead-time reductions can be obtained by using more than one designer. This can be achieved by breaking the component down into suitable pieces. Key features are identified and represented in a master geometry model, which all designers reference for the basis of their part.

Figure 4 shows how this works for a complex cast part. The master model is referenced by each sand core model, which is in turn referenced by the cast component model. Cascading the link functionality a stage further gives a finish-machined part, again linked to the master.

Using this arrangement, changes to the master can quickly be propagated to the core models and on to the finished part.

The final stage is to add component tooling into the multi-model process. Each core model (described above) would be referenced by tooling models, thereby enabling changes to be propagated to the tooling.

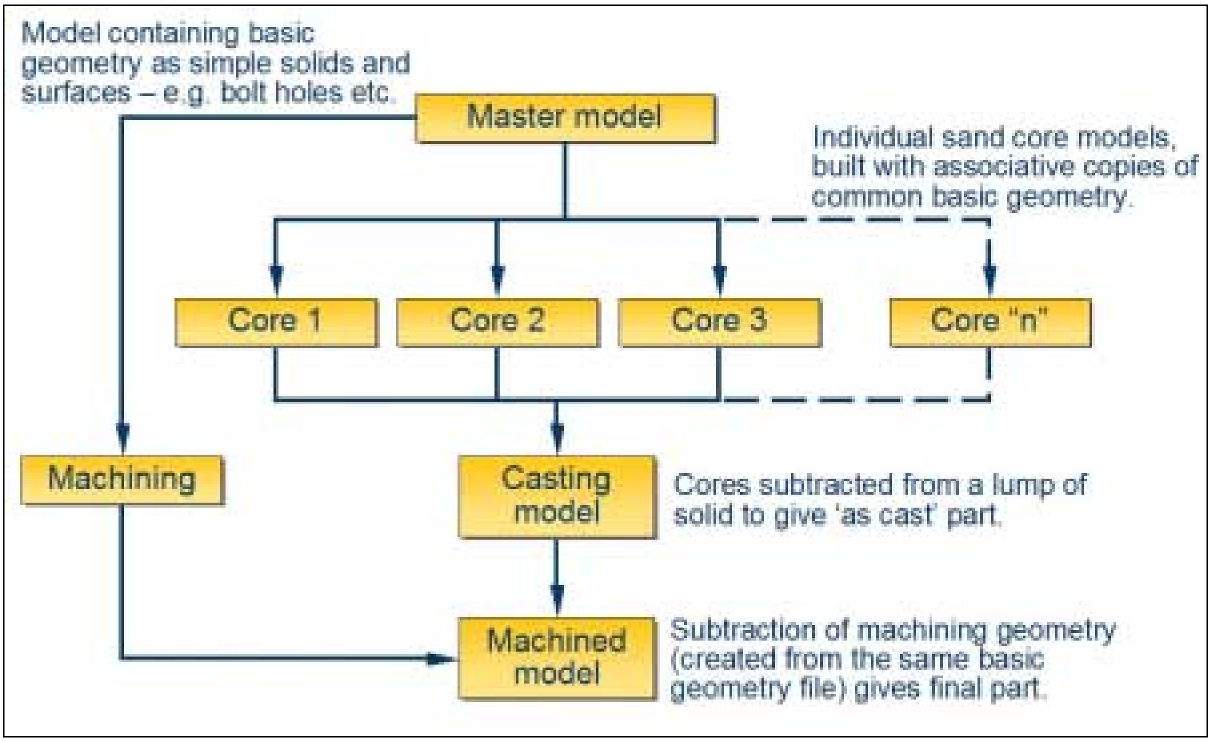

Figure 4 – Multi-Model Structure for a complex cast or moulded component

4.2.5 Two parallel process models for tooling

There are two process concepts currently in use in tooling design to support parallel working with CATIA. The choice of which approach to adopt will depend on business needs and customer-supplier relationships.

4.2.5.1 Design-biased approach

The 'design-biased' approach is used where companies want their own designers to carry out detail design/DFM, and retain close control of the component form. In this case tooling designers consult closely with the component designers to ensure suitable DFM features are included in the component. At the same time, the tooling designers work on the tooling, using linked copies of the design data. Design changes are thereby automatically propagated to the tooling.

4.2.5.2 Tooling-biased approach

If a company is willing to out-source the entire detail design/DFM of a component, a 'tooling-biased' approach can be used. The tooling supplier would be charged with modelling the component for the customer (often on the customers premises) while concurrently designing the tooling.

In this case the customer retains authority over the master geometry model and component form, but the underlying structure is defined and actioned by the supplier. In practise, the supplier creates tooling models from which component models can be produced for the customer with little extra effort. Tooling is consequently master, so the tooling models need to be modified to make a change to the component.

4.2.6 "Core and Cavity Design"

Core and Cavity Design is a CATIA V5 product aimed at the form tooling industry. Users can rapidly define the core and cavity sides of a part, including any sliders or loose cores that may be required. It includes functions that analyse the part for draft-ability and define the pulling directions, parting lines etc.

Powerful automated surfacing capabilities permit rapid manipulation of the component form to add draft, fillets and surfaces as required. Groups of surfaces are then automatically produced for each 'pull'

While primarily aimed at injection mould tooling, Core and Cavity Design also lends itself to casting or press tooling design where customers have supplied a part without correct production definition, perhaps from another CAD system.

4.3 Design validation

Electronic validation, performing tests or simulation on 'virtual' parts, is one of the major benefits of 3D CAD. CATIA V5 sets new standards in this area by providing a single, truly integrated environment for design and validation. It is now possible to perform part and assembly design validation as an integral part of the design process, rather than a post-design check. This applies to both Digital Mock-Up (DMU) review, and Finite Element Analysis of parts and assemblies.

4.3.1 DMU review capabilities

The most commonly used DMU review tools are for measurement, automatic clash/clearance detection, and dynamic sectioning. These tools are all available from the assembly design workbench and can be used interactively during product design.

Kinematics simulation is also available. Assembly constraints are used to help define the mechanism, which can then be articulated between defined limits. Complex sequences of movement can be built up to simulate a complete cycle of operation for a tooling assembly, for example. Interference between components can be detected and displacement, velocity and acceleration curves generated from the motion of specified parts. Being based on assembly geometry, changes are automatically accommodated.
Further geometry-based analysis is available in CATIA V5 in the form of 'fitting simulation'. The DMU Fitting Simulator allows the user to define and simulate assembly and disassembly procedures, thereby validating product assembly and maintenance at the design stage. An example is the assembly of sand cores into a core-pack. Given the start and end positions, CATIA V5 can automatically verify assembly is possible and calculate the best insertion trajectory. Should assembly fail, the user is able to visualise the problem areas.

4.3.2 Associative structural analysis

Finite Element Analysis is traditionally viewed as a time consuming and highly specialised task. This conflicts with the need for rapid response in the early stages of a design project, where meaningful analysis would be of great benefit in establishing design direction.

Software development has reduced the time and difficulty of building and running FE models. It is now possible for designers to use FEA as a concurrent part of the design activity to help converge on a valid solution more quickly, subject to suitable training and procedures being in place.

With CATIA V5 analysis products, design changes do not necessitate a new FEA model being built from scratch (as before), saving yet more time. Loads and restraints are applied to the CAD geometry, and update with the part when it changes; ideal in the early stages of a project. This applies equally to parts and assemblies. Only when the 'calculate' button is pressed is a mesh generated (automatically) and results calculated.

As the design progresses, more detailed FEA can be carried out by specialists. Should specialised software capability be required, 3rd party analysis packages are available which run seamlessly in the CATIA V5 environment (as an extra workbench), and use the native CATIA V5 CAD models.

4.4 The need for Product Data Management (PDM)

Business improvement through the use of CAD can only go so far before running into data management problems. The sheer number of CAD files created in concurrent engineering programs, and the need to prevent over-writing, re-work and the use of incorrect data all demand a means of management. Furthermore, investment in knowledge-based engineering processes demands a means of robust knowledge capture, storage and retrieval.

A PDM system is a tool for managing product definition information and the product development process. It provides a means of revision control and archiving as well as ensuring the right people can access the right data. Modern PDM systems also act as an enabling technology for concurrent engineering, allowing several engineering activities to be conducted in parallel.

Essentially PDM systems use database technology to manage data related to a product and the processes used to design, manufacture and support it until the end of its useful life. This may involve controlling product configurations, bills of material and associated design variations or versions, which change as the product moves through design, manufacture and service.

Data in a variety of file formats can be stored and controlled, including CAD models and drawings, NC code, analysis results, product specifications, bills of material, project plans and correspondence, for example. Powerful search and retrieval tools rapidly reduce the time taken to find and utilise items under PDM control, encouraging re-use of existing data where appropriate.

PDM systems are also used to enforce defined business processes. 'Workflow' is one name given to a PDM function, which moves data through an organisation in accordance with a pre-defined path. Data is routed automatically through a series of users. At each 'stop' the nominated user is informed of an action they are required to take before the data can be moved to the next stage. Typical examples are design sign-off or concession authorisation, but more complex processes can be managed this way assuring adherence to quality procedures and enabling progress tracking.

Operation with other engineering or manufacturing systems such as legacy PDM solutions, ERP and MRP systems can be achieved by using dedicated integration toolkits from the PDM solution provider.

As the backbone of a company's CAD installation, PDM technology provides significant potential for improving productivity and reducing development timescales and costs.

4.4.1 PDM and CATIA V5

CATIA V5 is designed to work seamlessly with ENOVIA and SMARTEAM PDM products, all members of the CATIA family. The main benefits of this tight integration surround the ability to manage advanced CATIA functionality from within the PDM system. Using other PDM systems risks limiting the potential benefits of CATIA V5 because they are incapable of managing CATIA-specific functions.

CATIA V5 techniques for tooling design allow associative tooling pack models to be created directly from a master data set (see section 4.2.4). This controls not only the component geometry, but also the tooling pack and any subsequent operations. To make this possible a succession of links are created between the models. These links need management and manual methods soon become unworkable as the number of models, assemblies, associative NC machining files and their links increase.

ENOVIA and SMARTEAM PDM products have the capability to manage these complex link structures and ensure that the user always has access to a full representation of the product structure.

Descriptions of the CATIA V5 PDM solutions follow, below.

- 'ENOVIA VPM' (Virtual Product Manager) is the existing CATIA V4 PDM system, used by major OEMs the world over, which has been developed to manage CATIA V5 data. Running on a UNIX server it is able to manage CATIA V4 and CATIA V5 data concurrently, thereby facilitating migration. Advanced functions include the ability to automatically flush design changes through a product structure (updating multi-model links), and remote management of assembly configurations.

- 'SMARTEAM' is a cost effective, Windows-based PDM system suitable for small to medium sized enterprises. In addition to standard PDM functions, CAD data can be inspected through an integral viewer and made available over multiple sites or the Internet. The system is licensed so it can grow with a business, and integrate seamlessly with a number of CAD systems and office software packages. CATIA V5 integration enables full part and assembly management and the maintenance of dependencies inherent in concurrent engineering processes.

- 'ENOVIA V5 VPM' is an all new, next generation PDM system designed specifically for CATIA V5 and related products. Available as a product in it's own right, it also forms the core of an integrated, enterprise-wide engineering, communication and management system which can be tailored to suit different businesses. ENOVIA V5 VPM has all the features of its predecessor and SMARTEAM, packaged behind a consistent 'web-style' interface.

4.5 Hardware and software

The CATIA V5 portfolio represents a collection of over 140 software applications. These have been written to run on both UNIX® and Windows™ platforms, and are sold in a range of configurations to match business needs. The following section discusses basic hardware requirements and introduces the way CATIA is packaged, licensed and sold.

4.5.1 Hardware

CATIA V5 is written for both Windows™ and UNIX® operating systems. A company's choice of platform will depend on existing hardware, the need to run or work with other applications and how demanding the use of CATIA V5 is likely to be. Mixed installations are possible, perhaps for companies needing to make a phased transition from UNIX to new Windows infrastructure, provided older UNIX machines are suitably specified.

CATIA V5 performance is heavily dependent on processor speed, memory and graphics capability. A hardware approval scheme is in operation to certify new computers for use with CATIA V5. Purchasing supported machines is a valuable means of minimising the risk of problems, especially given the variety of hardware on the market.

The advantages of operating a Windows-based infrastructure include significantly lower hardware cost (including support), availability of administration skills and compatibility with other business software.

UNIX systems are a proven, stable platform and provide an 'open-source' operating system allowing customisation. Furthermore their architecture is built around parallel processing which is particularly suitable for demanding computation such as FE Analysis.

4.5.2 Software

To simplify the supply of CATIA V5, the range of applications has been broken down into 'Platforms', 'Configurations' and 'Products', described below.

4.5.2.1 CATIA V5 'Platforms'

The applications in the CATIA portfolio have been split into three platforms, akin to trim levels on a car. They are termed P1, P2 and P3.

- P1 level products are aimed at enterprises working on low to medium complexity products. Functionality and cost compares favourably with competitor products, and CATIA V4 data can be worked with.
- The P2 platform represents the market previously serviced by CATIA V4, typically medium to large sized enterprises, or those with demanding CAD

requirements. Migration from CATIA V4 is supported with extensive V4 interoperability tools, and full translation capability.

- P3 products targets low volume, high complexity specialist requirements such as class 'A' surfacing and Aerospace Sheetmetal Design.

Data is interchangeable between platforms meaning a company can run a mixture of P1, P2 and P3 seats if required

4.5.2.2 Software configurations

"Part Design 1", "Assembly Design 1" and "Generative Drafting 1" are CATIA V5 "products". Commonly required groups of products have been created called "configurations". These form the basis of most companies' installations.

Should extra products (not included in a standard configuration) be required, they can be added to the license.

4.5.2.3 Licensing

Software licenses can be fixed (node-locked) to a particular machine or floating (served) across a network, enabling access by multiple users. In the case of floating licenses a computer needs to be configured as the license server. Machines requiring access to floating licenses must be configured as 'clients' to enable communication with the license server.

4.6 Training and support

Optimal performance requires well-trained users using best-practice methodology and process. Initially these are unlikely to be provided solely from within the organisation and even in the longer term, a relationship with a suitable consultant may prove a valuable way of encouraging continuous improvement into the future. This type of service has been provided by Integral Powertrain to a number of customers with great success.

4.6.1 Basic training

CATIA V5 has been developed with a strong focus on 'ease of use'. Anyone with experience of Windows ™ based applications will find CATIA V5 instantly familiar (whether it's running on a PC or UNIX platform), and those with knowledge of any current 3D CAD system should be able to make rapid progress.

With CATIA V5, the time taken for the average user to reach a good level of proficiency has been dramatically reduced. The user interface has a consistent 'look and feel' across all workbenches, which helps users get up to speed quickly, and commands follow similar patterns, giving new functionality a familiar feel.

As a guideline, *basic* training for an operator experienced in another 3D CAD system will require approximately 5 days (compared to at least 10 for CATIA V4). It would be wrong however to suggest that this implies that the total amount of training required should be reduced for the reasons described below.

4.6.2 Methodology training

CATIA V5 is a powerful product, but obtaining optimal performance is very dependent on methodology. Training plays an important part in rapidly reaching this level and covers a wide range of requirements, from learning how to use particular functions, to building change tolerant parametric models, to implementing complex parallel working structures.

Clearly such methodology training must be provided by trainers who are experienced in the latest techniques and their application in real business environments. This may challenge traditional training sources and is likely to link methodology training with process improvement. (see section 4.7)

4.6.3 Process development

Developing template models and process tools (similar to those described in section 4.7) requires input from all areas of the business (including suppliers). These inputs need to be distilled into coherent processes, which can be managed by the company PDM system and actioned through CATIA and other appropriate tools.

Defining a process, it's inputs, outputs and dependencies is in itself a specialised task, which may require training or the help of a consultant. Coding a process within CATIA V5 will definitely require advanced training, or the employment of suitably skilled personnel.

If a company is moving towards heavy reliance on automated engineering processes, a department may need to be set-up to maintain and develop them. The cost of running such a group is likely to be small compared to the savings in engineering effort realised across the business.

4.6.4 Ongoing CAD review

The CATIA V5 family will continue to grow. Products, processes and enhancements will become available to service requirements identified by an ever-growing body of users, across a wide range of industries.

It would be prudent to perform some form of ongoing CAD review, to gain awareness of advances that may offer the potential for significant business benefits. New functionality will need to be thoroughly tested, and the impact on existing processes closely studied. Implementation and update training will also need to be managed.

This is another area where the services of suitable consultants may be more effective than in-house resource, in terms of both cost and expertise.

4.7 Continuous improvement

In addition to general functions, CATIA V5 supports a number of advanced technologies designed to make the design process more efficient. Making use of the methods described below, a business can formulate a policy of continuous improvement, which simultaneously encompasses all areas of product development.

4.7.1 Knowledge-Based Engineering (KBE)

Knowledge-Based Engineering is a technology for capturing best practise and re-using it in a new context. This already goes on at a human level, with people learning from what they did on the last job, but in CATIA V5 terms it applies to design rules and processes which can be formally captured by hard-coding into a CAD model for re-use in future projects. The CATIA V5 tools to facilitate such activities are collectively termed ‘Knowledgeware’.

4.7.2 Additional tools for the designer

Simple knowledgeware functions are available to help a designer while creating geometry. For instance, a designer can set checks to warn if a geometry change causes a part to go out of specification in a defined area. Perhaps a tool becomes too heavy to be lifted by a particular machine when a dimension is changed? In this case a ‘rule’ could also be set to automatically adjust another area of the part to adjust the weight back to a required limit.

Real-time interfaces with external software (such as spreadsheet programs) allow designers to link their own computer-based design and review tools with CAD models. Existing tools, which may previously have needed manual data input can now be linked to live CAD data.

4.7.3 Re-using information – template features

Knowledgeware effectively turns CATIA into a programming environment, allowing ‘super-users’ to build models with the ability to adapt to different inputs or circumstances. These models can then be put to general use to rapidly create parts and assemblies. This is the basis of process ‘templating’.

The foundation of this technology is the ability to change the references, or starting points that a model subsequently performs operations on, or adds to. If the only things that change time after time are the references, then it should be possible to automate the remainder of the operations, for re-use on other projects.

A basic example of the use of Knowledgeware, as applied to templating, would be a casting core-print feature whose form changes within strict guide-lines, depending on the exact context in which it is used. Using the CATIA V5 Product Knowledge Template functionality, it is possible to create a model of such a feature with in-built code controlling what form it takes when inserted into a model, depending on the geometry (or references) used to position it. With a library of commonly used features like this, all designed to adapt to the particular context they are inserted into, significant time and quality gains can be made.

4.7.4 Template models

The next logical step is to build whole component models that adapt to particular requirements, and produce associative tooling models. CATIA V5 has a number of functions specifically aimed at making models 'change-tolerant', vital if a component's topology needs to vary significantly.

Once created, the template model contains the building blocks for a variety of design iterations. By working through a structured series of questions, driving dimensions and information for the template model are automatically extracted from reference documents (standards, company records, parts database, etc.). The template and linked tooling models are adjusted and the resulting collection of models, although basic, meets all specified requirements.

Template models should capture best practise and assure consistency of design. Furthermore, once in place it is only necessary to validate and maintain one approach.

4.7.5 Business templates

Taking the template concept a step further, whole processes can be automated, allowing the establishment of the product structure, information flow, relationships and deliverables at a high level. Distilling the business/project level workings into 'templates' allows them to be rapidly deployed in the future, and facilitates process visibility and control.

Fundamental to this approach is an understanding of the rules that govern the creation of the product at all stages, and the information flow, both within the company and with suppliers. It is then possible to automate the geometry creation in line with known processes and ensure that all factors affecting the design and manufacture of the product are considered. This can be either through interactive forms or through automatic reactions to the inclusion of particular features.

The CATIA Business Process Knowledge Template (BKT) supports this way of working, by allowing companies to capture their business processes, and to share and deploy them through their own custom CATIA V5 workbenches.

Experts use the BKT authoring tools to automate design and engineering tasks in line with the methods described above, embedding core-knowledge rules and checks. The result is a custom-built CATIA V5 workbench for each business process.

The end user simply accesses the required BKT custom workbench then follows the structured process to arrive at a solution.

4.8 Migration

Changing CAD and PDM systems may require changes to hardware and network infrastructure, as well as introducing issues of data compatibility between new and legacy CAD systems.

A thorough understanding of the entire migration process should be attained before a migration plan is formulated. Areas to consider should include:

- Suitability of existing hardware and software for use with new systems.
- Impact of the migration on communication in the business and with suppliers.
- Disruption to the business at all levels.
- Contingency for unexpected problems.
- The need for specialist support to make changes to existing systems.
- Hardware support contracts.

Businesses planning to move to CATIA V5 must define a strategy for the management and continuing use of legacy data. The migration of legacy data management systems must be considered as part of the new PDM solution. The method by which the data is made available in CATIA V5 however, is a separate issue, which depends heavily on the legacy data format and what level of operability is required in V5.

The strategy for CATIA V4 users is helped by extensive V4/V5 interoperability tools, and differs from those coming from other systems, for whom a number of different options exist. The key choices and their relative merits are discussed below, but in summary they are as follows:

Migration from CATIA V4:

- Work with CATIA V4 data, and only migrate data to CATIA V5 if it needs to be modified
- Migrate data to CATIA V5 without history
- Migrate data to CATIA V5 with history

Migration from other CAD systems:

- Translate via specialist software
- Translate via neutral format (STEP/IGES)
- Purchase CATIA V5 ‘plug-in’ to use legacy data in assemblies (read only)
- Use CATIA V5 ‘feature recognition’ to build history

4.8.1 Migration from CATIA V4

CATIA V5 can read, review and work with CATIA V4 data (including drawings), without the need to perform any conversion. Companies with a vast collection of CATIA V4 parts that are never going to be modified will consequently not need to convert them to the CATIA V5 format.

The tools for working with CATIA V4 models in CATIA V5 are well developed and very reliable. Many CATIA V4-based companies are already using the CATIA V5-family of DMU (digital mock-up) tools in place of CATIA 4D Navigator (the V4 DMU software), as a pre-cursor to full migration to CATIA V5.

CATIA V4 parts that do need to be modified in future can be migrated to CATIA V5 in two ways. The first method is extremely reliable, but gives a part with no history. The second method yields a CATIA V5 part with full history and is generally successful, but can fail if the CATIA V4 model is poor. In such cases, remedial attention to the CATIA V4 model should assure success. There are CATIA V4 model checking facilities built into CATIA V5, which are useful when the migration to CATIA V5 has commenced, but companies beginning to plan for migration may wish to take steps to assure themselves of their CATIA V4 model quality while there is still time to do something about it. This is an area where the use of 3rd party checking tools, which verify each CATIA V4 model complies with a set modelling standard, can prove its worth.

The key to a successful and cost effective transition from CATIA V4 therefore, lies in using an appropriate blend of CATIA V5 migration and native CATIA V4 data.

4.8.2 Migration from other systems

In the first instance, a business should consider the level of operability required from the use of legacy data in CATIA V5 (i.e. can models be translated from other CAD systems with history etc.?). In simple terms, the more operability required, the higher the cost. A migration policy should be formulated to establish effective solutions to meet realistic requirements.

4.8.2.1 Specialist translators

Specialist translation software can offer history-supported migration in some cases. This solution produces models that can be edited (to an extent depending on the specific software), and can ultimately save time and money if re-modelling legacy parts represents significant effort.

4.8.2.2 Translation via neutral data formats

Models can also be converted to CATIA V5 format by using STEP or IGES translation, both well established, industry standard neutral data formats. STEP is the recommended neutral format, because of its reliable solids conversion and robust product structure recognition capabilities. This allows assemblies or product structures sent from other systems to be correctly converted into a CATIA V5 equivalent.

4.8.2.3 Re-building history with CATIA V5 'Part Design Feature Recognition'

Solids resulting from STEP translation have no history, but in most cases it is possible to re-create at least some history using the CATIA V5 'Part Design Feature Recognition' product. Users identify the surfaces associated with a feature, like a fillet for example, and instruct the software to recognise them as such. By identifying each feature in turn, a history tree can be constructed for subsequent editing.

4.8.2.4 Using legacy data without translation – CATIA V5 'Multi-CAD Plug-ins'

The final method of working with legacy data is by means of 'MultiCAD Plug-ins'. These are akin to specialist translators that are well integrated into CATIA V5 and the legacy system. It is possible to automatically create CATIA V5 versions of legacy parts (without history) or to produce tessellated versions, which can be used in the same way as native CATIA V5 parts in 'visualisation mode' (as introduced in section 4.2.2). This means DMU review functionality is supported, and hybrid assemblies containing both CATIA V5 and legacy data can be created.

Multi-CAD Plug-ins are available for most major CAD systems and offer an effective means of enabling migration, and supporting multi-system working in the transitional phase of migration.

5 Case Study

5.1 Introduction

Company 'X' designs and manufactures assembly fixtures for the automotive industry. Its customers are generally automotive OEMs. Some basic statistics are given below.

Business	Assembly fixtures
Turnover	**€19M**
No. of employees	**120**
No. of design engineers	**48**
No. of production engineers (N/C programmers)	**6**
No. of production workers	**28**

The company is finding itself increasingly in competitive tender situations. This is not only placing pressure on margins, but also affects other aspects of business as competitors try to make themselves more attractive to customers. In particular, lead-times in the sector are reducing rapidly and this is leading to a number of problems related to efficient scheduling of machining and engineering work and effective project management and change control.

The management team recognised that the capabilities of CAD systems were increasing and decided to carry out a pilot study to identify how these might support business level improvements. The study was carried out over a 3-month period with the assistance of a CAD systems consultant, expert in the application of CATIA V5.

Firstly, the high-level business objectives were identified. Next, a thorough review of existing practice was carried out and potential areas for improvement identified. Key process improvements were then validated on the new software, the results forming the basis of a Return On Investment (ROI) calculation.

5.2 Business objectives

The management team had agreed on the following objectives:

- ROI in less than 3 years based on improved efficiency.
- Improve quality of tenders within same time and resource constraints.
- Be best in class at customer communication. This entails communicating more quickly and professionally with engineering, project management and purchasing areas.
- Develop dependable, cost efficient outsource options to cover demand peaks.

5.3 Business process review and improvements

5.3.1 Bidding

Most new opportunities contain a tendering phase. Some tenders relate to 'commodity' items such as manufacture of new or replacement items to a given design. For this type of project the tendering process can be very short. In order to be successful it is essential to be able to work out quickly and accurately the required resources. The company also wished to improve its capability to locate out-sourced machining capacity and negotiate favourable terms. These requirements were considered during the review of design, resource planning and purchasing activities.

Larger opportunities often require much more complex tenders during which the company develops a concept scheme. The detail design would later be produced from this concept. A more mature concept design was considered to be a great advantage in this type of bid as it enables more accurate cost analysis and gives the customer greater confidence.

5.3.2 Design

Although the products are bespoke, many of the operating principles and mechanisms are made up of similar elements. Given this similarity particular attention was paid during the pilot study to investigating 'smart' design tools, and methods of design automation.

Also the potential benefits of 'Assembly Design' and 'Multi-Modelling' were investigated with the objectives of improving efficiency and reducing the impact of design change.

Total amount of design cost (labour only):

(Note: The number of man-hrs/year/employee is assumed to be 1700 hours.)

48 designers x 1700 x €50/hr = Annual cost of **€4,080,000**

This breaks down as follows:

Discipline	Percentage of total design resource	Cost
Concept Design	10%	€408,000
Detail Design	40%	€1,632,000
Design Changes	20%	€816,000
Drawing Creation	30%	€1,224,000

5.3.2.1 Concept design

The primary objective was to improve the quality of design schemes given the same timing and resources. This was considered essential to win work and maintain the business. During the pilot project CATIA V5 assembly design functions demonstrated the potential for substantial efficiency increases. It was also established that the use of parametric templates for certain features would result in further improvements.

Concept design represented 10% of the total design activity. Following the pilot study it was considered that the introduction of CATIA V5 would provide a 30% improvement in the level of detail achieved for the same level of resource. This supports the requirement to improve bid quality in line with the business objectives.

5.3.2.2 Detail design

40% of the total design activity was associated with the creation of the initial design data. The introduction of data re-use, template models, libraries, design tables and the application of knowledge-based engineering resulted in a 20% reduction in design effort.

Cost of detail design activity per year = 40% of total design activity

40% x €4,080,000 = €1,632,000

20% reduction in effort, equating to €326,400

5.3.2.3 Design changes

The main drivers of design changes were found to be:

- Customer driven specification change
- Design failed to meet validation criteria
- Late input from manufacturing source

Often changes knocked on into many components and caused substantial delays.

20% of the total design activity was found to be associated with supporting these design changes. During the pilot project it was established that CATIA V5 provided opportunities to substantially reduce this figure. Firstly, by using associativity and multi-modelling techniques, the effects of changes could be cascaded automatically through the assembly. This can reduce the impact of a change substantially. Secondly, through the use of smart design tools, design rules could be introduced. These reduce the chances of failing to meet validation criteria. Finally, by sharing the more detailed concept schemes with manufacturing engineers, key requirements could be highlighted before detail design was undertaken. It was estimated that these improvements would result in a 25% reduction of the design effort relating to design changes.

Cost of change activity per year = 20% of total design activity

20% x €4,080,000 = €816,000

25% reduction in effort, equating to €204,000

5.3.2.4 Drawings

With the time taken to produce drawings, and the increasing use of full CAD/CAM manufacture, the company was keen to start the transition towards drawing-less design.

Drawings represented 30% of the total design activity. Annotation of models with the necessary manufacturing details reduced the requirement for drawings and allowed drawing-less production for in-house manufactured components (outside suppliers not as yet being capable in this area). This resulted in a 15% reduction of design effort.

Cost of drawing activity per year = 30% of total design activity

30% x €4,080,000 = €1,224,000

15% reduction in effort, equating to €183,600

5.3.3 Validation

Historically much of this activity had been done in hardware (trial builds, clash checks through to functional tests). This was still the main way that confidence was achieved although the requirement for reduced lead-time was found to be putting a great deal of pressure on this area. The company realised that failure to carry out sufficient validation was likely to result in very serious problems, possibly even in delays to OEM production schedules. They determined therefore that the only satisfactory solution was to develop robust 'virtual' validation techniques.

During the pilot study it was found that many of the hardware validation activities could be carried out virtually in 'Assembly Design', or with CATIA V5 DMU products. This included fitting simulation, kinematics and clash detection. The company therefore decided to use experienced engineers to define a validation plan making use of these tools.

The status of components and systems could also be monitored much more rigorously through the PDM system. As an example, component release level was linked to signing off (electronically) to record that appropriate validation activities had been carried out.

A budget figure of 5% of total design activity was allocated to developing and carrying out virtual validation. It was assumed that in the first year this would be an additional cost but that in subsequent years it would be recovered by reduced hardware based validation.

Cost of virtual design validation = 5% of design activity

5% x €4,080,000 = €204,000

5.3.4 Finite Element Analysis (F.E.)

In some cases, the optimisation of a particular area is important for an efficient design. Inadequate rigidity compromises function whereas excess mass may affect technical performance and drive excess cost. Prior to the pilot program the company had tried with limited success to introduce FE analysis for this type of problem. This was based around the use of a proprietary analysis package. Data was transferred from the CAD system before being meshed and solved using the analysis software. Unfortunately by the time the analysis was completed, the design had often changed or been frozen. Also, the possibility of using analysis as an optimiser was compromised by excessive cycle-time.

During the pilot study it was found that CATIA V5 analysis tools could be of great benefit in this area. The mesh generation tool was found to produce acceptable mesh quality in much less time than the existing process. This could be exported to the dedicated analysis package if required, although for many problems the structural analysis tools provided in CATIA V5 were found to be adequate.

It was decided that analysis procedures should be produced to enable trained designers to carry out routine analysis and optimisation tasks within the CATIA V5 environment. Experienced analysts would supervise these activities and carry out complex or non-standard tasks.

It was anticipated that the analysis team would have to create procedures for the design engineers; therefore initially there would be no saving in resource. However once the basic analysis procedures were in place and had been validated, it was expected that the amount of support required by the analysis group would be less and the number of specialist analysis engineers could potentially be reduced.

Additional recovery is anticipated thereafter through reduced hardware-based validation.

Cost of analysis procedures = €83,000

5.3.5 Manufacture

The manufacturing department was already making use of CAD models and NC programming to drive machining centres. However, efficiency gains in terms of reduced manpower and shorter lead-times were sought through the use of a single CAD/CAM data format.

5.3.5.1 N/C programming

The existing software used for NC programming was a third party product and relied on the use of IGES/STEP translations. The pilot study showed that the CATIA V5 software offered the required functionality provided by the existing software but avoided the need for data translation. In addition, by using a single product for both geometry creation and manufacturing, the associative tool paths could be updated inline with changes to the geometry. This offered the potential to reduce the workload on the NC programmers and the time to implement change.

For the purposes of ROI the company decided to ignore any benefits resulting from the use of CATIA V5 in the area of manufacturing, however the belief is that the overall process is far more robust since the company has been operating on a single software product.

5.3.5.2 Inspection

There are plans to drive the CMM machines from master CAD data, and this is required to support the desire to reduce the need for drawings. To date this area has not yet been resolved and remains a development area for the future.

5.3.6 Commissioning

On large tooling assemblies, the structure is assembled and commissioned on-site at the client's production facility. Supporting documentation and communication is usually required to support this activity at considerable expense.

The use of CATIA V5 has allowed designers to quickly create assembly documentation, exploded views and bills of material, which can be used by the engineer during the assembly process. This was not possible before because of the complexity of some tooling assemblies and limitations with the previous CAD system.

Improvements in this area were ignored for the purposes of the ROI study, although benefits are expected when current projects reach the installation phase.

5.3.7 Maintenance and refurbishment

On large tooling assemblies the company often provides a lifetime service support contract. This requires a process that can identify all the components within an assembly and provide the necessary information to allow the components to be replaced. Currently this information is held in a number of ways that are not considered to be robust.

It is intended that in the future this will be handled by the PDM system, which will control issue status, product configuration and management of associated documents.

This was omitted from the ROI study.

5.3.8 Purchasing

Short lead times are increasingly driving short-term peaks that place a lot of pressure on internal manufacturing capacity. Increasingly machining activity is being out-sourced to cover these peaks. The Purchasing department are responsible for co-ordinating this activity and wished to streamline the process. The result is the development of an 'outsource pack' designed to provide all the information required by a supplier to make a rapid and accurate bid and to carry out the task if successful. In this way they expect to find excess capacity at reduced rates. As confidence is built in co-ordination of out-source work, it is also intended to explore lower cost machining options in developing markets.

Some of this information is also used to produce a simple cost model for internal use (e.g. quantity of material and machining operations required). This enables estimates to be quickly provided to support bidding activities.

5.3.9 Project monitoring

The application and use of digital mock-up has allowed people that are not directly involved with the engineering to assess the maturity of the design. Improved project reporting has enabled data on project spend versus project maturity to be compiled, and plans have been made to implement Earned Value Analysis in the near future.

The size of the commercial team consisted of 16 people including sales engineers, project managers, account managers, purchasing and senior management. The additional resource associated with providing the capability outlined above was estimated at:

3,800 man-hours

3,800 hrs x €50/hr = an annual cost of €190,000

The benefits that have been achieved as a result of improved project monitoring have not been quantified in this business case since the benefits will only be seen in the long term.

5.3.10 Additional Costs

5.3.10.1 Hardware

Initially it was planned to run CATIA V5 on the existing UNIX hardware however it was discovered that the maintenance cost on the UNIX hardware was significant and increasing due to the age of the equipment. For a modest investment the UNIX workstations could be replaced with the latest Windows-based hardware, which offered greatly improved performance, and was supplied with a 3-year on-site warranty.

Hardware investment:

54 workstations, server & installation	€127,000
Existing hardware maintenance (p.a.)	€70,000
Total additional expense	€57,000

5.3.10.2 Software

The overall software cost was €875,000, which was based on a core configuration of solid modelling, surfacing, detailing and digital mock-up. In addition there were floating licenses consisting of fitting simulation, kinematics, rendering, structural analysis, machining, sheet metal, core & cavity design and advanced knowledge-based engineering.

The software also included "SMARTEAM" Product Data Management across the whole company (including non-engineering departments) based on 75 seats. This was a prerequisite to enable the business to operate efficiently and maintain data consistency.

5.3.10.3 Software maintenance

No additional on-cost resulted from the new software since the maintenance charge was effectively the same for both old and new software solutions.

5.3.10.4 Training

Training courses were configured in conjunction with the CATIA V5 consultants to ensure consistent methods across the company. A basic training course of 10 days duration was provided covering basic operation and approved methodology. Further training was provided on selected areas of functionality to suit individual's needs.

The total cost for training and "at elbow" support was €111,400.

5.4 Return on investment

Additional costs incurred through new CAD system:

Activity	year 1	year 2	year 3
Project Monitoring	(€190,000)	(€157,000)	(€135,000)
Detail design (new)	€326,400	€335,500	€350,000
Detail design (changes)	€204,000	€210,000	€220,000
Drawings	€183,600	€188,500	€197,000
Design validation	(€204,000)	(€166,000)	(€50,000)
Structural analysis	(€83,000)	(€83,000)	€71,500
Software	(€570,000)	(€305,000)	-------
Hardware	(€39,000)	(€18,000)	-------
Training & Consultancy	(€111,400)	(€50,000)	(€23,000)
Total	**(€483,400)**	**(€45,000)**	**€630,500**

Pay back 2.1 years

6 Conclusion

The introduction of CATIA V5 enables companies to maximise the benefit of 3D CAD across the business. Real competitive advantage can be gained through concurrent working and efficient data management, and ongoing improvement is supported through integrated products that promote design automation and knowledge-based engineering.

CATIA V5 represents a step forward in the application of computer-based design tools to improving real-world engineering processes.

Key CATIA V5 capabilities for use in tooling design include:

- Robust and proven methods for concurrent working
- Built-in ability to support and automate engineering processes
- Fully integrated data management solutions
- Powerful yet easy to use design tools
- Extensive designer-level design automation capabilities
- Tools for working with data from CATIA V4 and other systems

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