APPENDIX C
Review of the CATIA V5 Software

The following is a summary of the capabilities of the commercial software product CATIA V5 (version 5) by Dassault Systemes. Of specific interest was the possibility of creating Knowledge-Based manufacturing models for arbitrary composite structures (as demonstrated in Section 5.4 of this thesis) since this class of problem is indicative of the current limitations of CAE software for composite structures. It was found CATIA V5 is unable to achieve this level of flexibility due to the fundamental nature of its software architecture.

C.1 Methodology of Knowledge Capturing

The tutorial documents packaged with CATIA V5 discuss the meaning of the term 'Knowledge' that was adopted in the system. The specific definition used by the software product's creators was as follows:

"Understanding how a product is intended to function and how to apply that understanding to create an appropriate product is knowledge"

CATIA V5 captures this knowledge in parameters (intrinsic or user-defined types are available) and relations (formulas, design tables, rules and checks). All product knowledge is easily accessible from the 'Specification Tree'. This is a hierarchical tree view display that expands and contracts (like the component in Microsoft Windows Explorer). Special filters can be used to control the nodes that appear in the Specification Tree since this is a convenient way to overcome information overload.

The various relations that form the product's knowledge are explained below.

C.1.1 Rules

In CATIA V5 rules are edited as Visual Basic scripts using an in-built editor. Each rule is automatically evaluated after a change to the design's parameters is made. Rules can apply to all design features of a given type or to specific features and are limited to have linear program flow (ie. no jumps or loops). Rules can alter other parameters of the model and hence trigger other rules. Also, rules can invoke macros that create extra features in the model, change parameters, get variables from external design tables, etc.

C.1.2 Checks

Checks have similar restrictions to rules regarding algorithm flexibility and are used to incorporate knowledge about the feasibility limits of a design's parameters. They are evaluated directly after the evaluation of all rules and any violations are reported to the user with a dialog box. The dialog box displays a user-defined text message for describing the nature of the check violation and/or suggesting changes to make the model feasible.

C.1.3 Macros

CATIA V5 uses macros that are stored in Visual Basic script files. They can be captured (recorded) directly from the user's interactions with the GUI. Also, they can be edited and replayed to perform common modeling operations or optimization. Macros are flexible enough to automatically create extra model data, extra rules, extra parameters, etc., through utilization of external data files.

C.1.4 Design Tables

These are used to manage a family of similar product designs (e.g. a parameterized set of configurations). Design tables are effectively Microsoft Excel spreadsheet documents that are closely linked into the CATIA V5 system.

C.2 Other Features

CATIA V5 includes an in-built optimizer for continuous variables. This is used to optimize a user-specified parameter that can be a function of user-defined rules. The user also indicates the free parameters for the optimization (e.g. lengths, thicknesses, etc.). There is a choice of 2 optimization algorithms - steepest descent or simulated annealing, and a choice of termination criterion. One powerful feature is the option to call CATIA's in-built FEA package called ELFINI during the rule evaluation. This allows the system to perform FEA based shape optimization in a manner similar to conventional structural optimization packages such as ANSYS (see Chapter 4), or GENESIS (see Appendix A).

Additionally, CATIA V5 supports user-friendly NC machine programming and outputs industry standard NC machine command files. The ordering of machining steps can be readily altered in the specification tree. Unfortunately, there is no scope for modeling more arbitrary manufacturing scenarios (e.g. composites) with this functionality.

A number of add-on modules would be needed to apply CATIA V5 to composite design tasks. However, they don't exist on the standard version under discussion here. These are presumably quite expensive but may facilitate and/or automate many of the tasks in day-to-day composite part design, analysis, etc. The following two modules appear to hold the most promise:

- *CATIA Generative Composites Covering:* Defines the composite part from an FEA point of view.

- *CATIA Generative Tape Design:* Includes ply features, ply properties optimization from contour information, true fibre path analysis, manufacturing process support, laser projection system programming.

C.3 A Test of Flexibility

An attempt was made, within the CATIA V5 environment, to model custom manufacturing processes by linking to an external costing database (as per the Process Link software reported in Section 5.4). Figure B.1 shows a screen capture of a simple rib model that was made in the CATIA V5 environment. In this model it was possible to define custom parameters (e.g. 'part area', 'part perimeter') and custom formulae (e.g. process time evaluation).

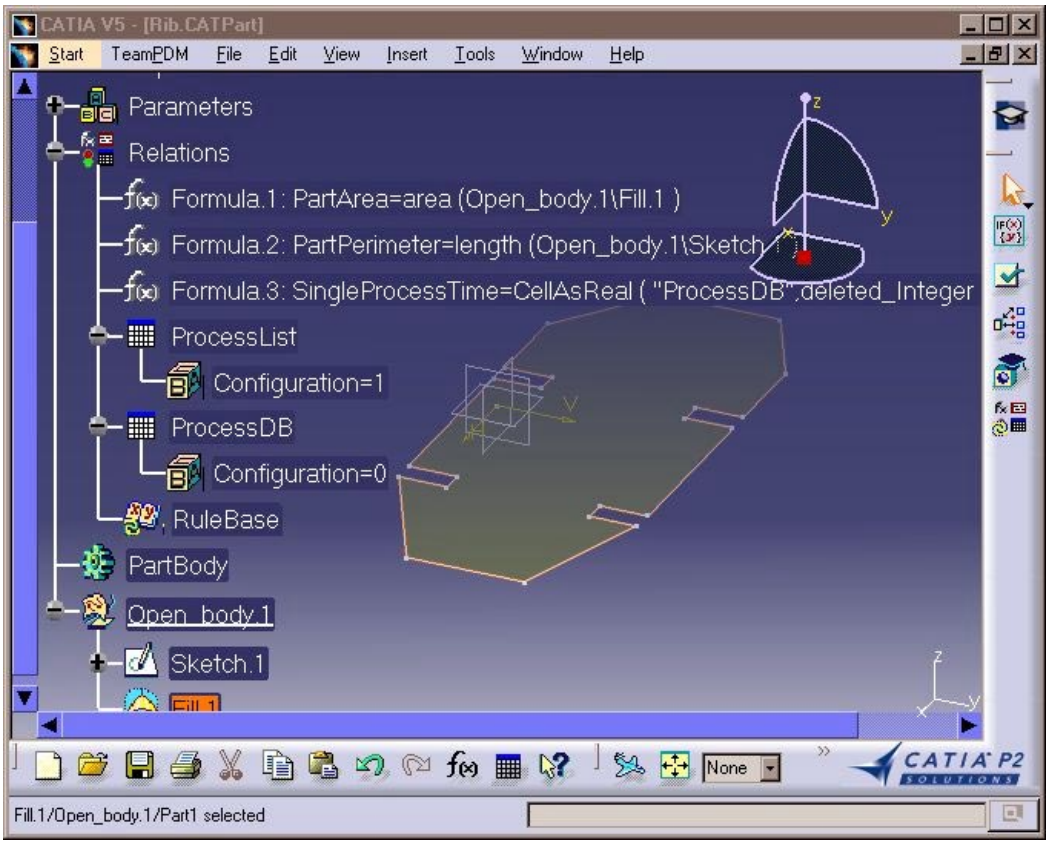


Figure C.1, Screen capture of a simple rib model in CATIA V5

The Specification Tree is visible to the left of the screen, super-imposed on the geometric model.

Figure C.2 is shows the CATIA V5 GUI being used to link a length parameter (the perimeter of the rib) to a formula. This was used to automatically calculate the value of a custom parameter designated ‘PartPerimeter’. The parameter could be a quite complex function of other parameters or part properties and it could also be subjected to constraints. This type of functionality is what sets CATIA V5 apart from many other CAD systems.

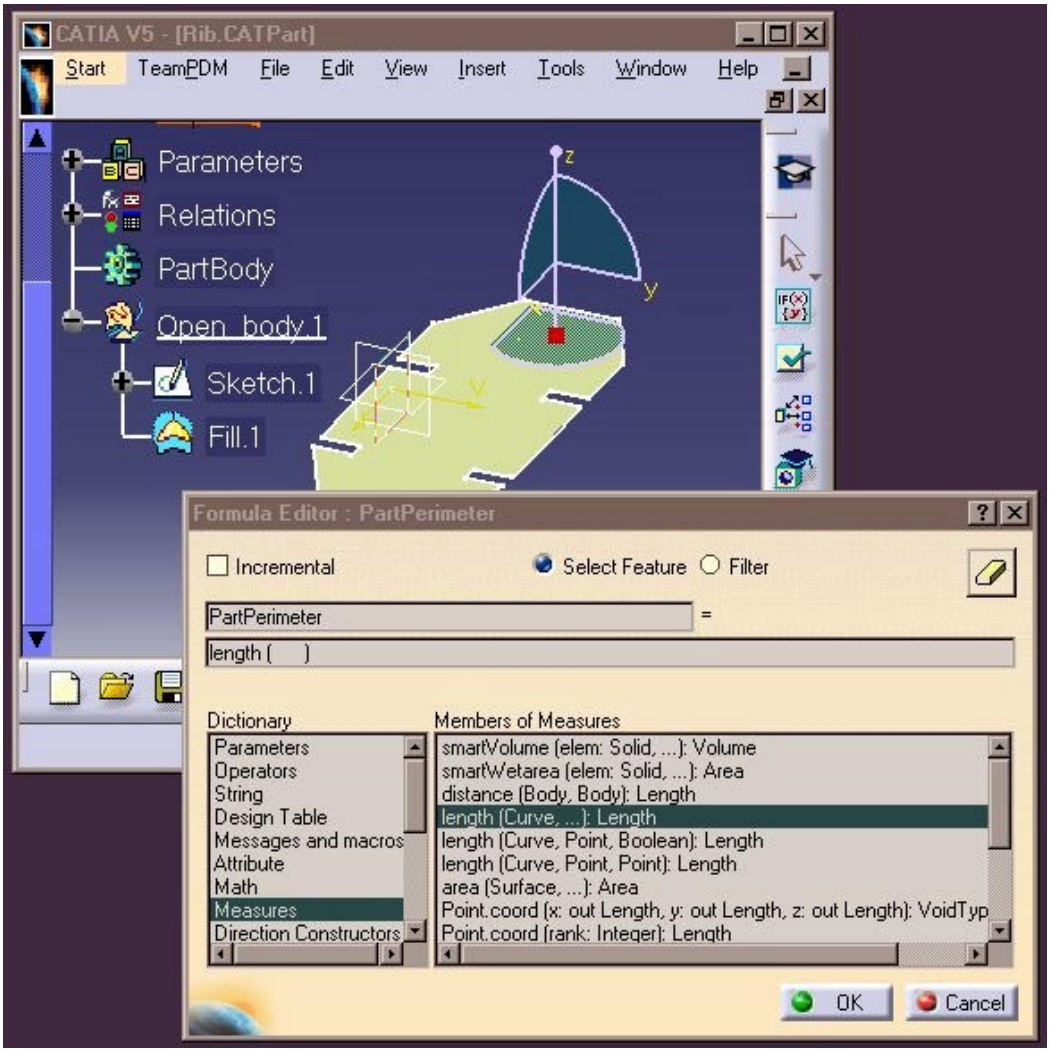
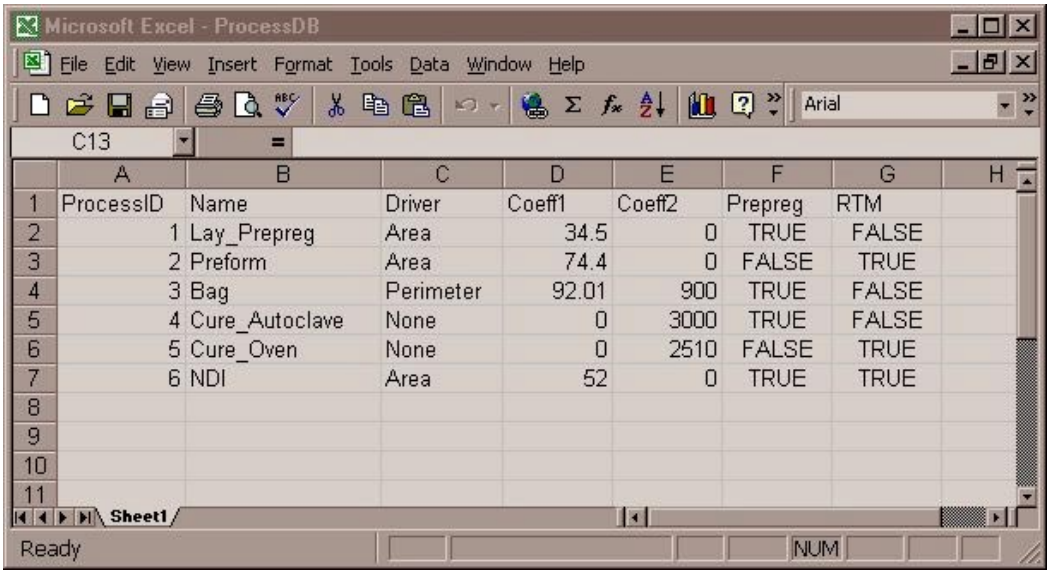


Figure C.2, Screen shot: linking geometric data into a custom parameter

Also, it was possible to link an external Microsoft Excel document of process time parameters into the environment. However, once linked, there were not many options since CATIA V5 treats all such documents as ‘product families’. Process lists and cost equations are different types of tabulated data that CATIA V5 cannot manipulate in a flexible way.

Given that CATIA V5 could not perform calculations of arbitrary information like process steps directly, it was attempted to ‘trick’ the software into evaluating a process model time estimate by treating the process steps as rows of a Design Table and sequentially summing the process times with a user-defined macro. Figure C.3 shows a Microsoft Excel document of pseudo-processes that was linked to the CATIA V5 environment. When linking to this table, the software could only examine one row at a time since it considers all Design Tables to be product families. However, the entire table should be considered simultaneously since many process steps are needed to produce a given product.



	A	B	C	D	E	F	G	H
1	ProcessID	Name	Driver	Coeff1	Coeff2	Prepreg	RTM	
2	1	Lay_Prepreg	Area	34.5	0	TRUE	FALSE	
3	2	Preform	Area	74.4	0	FALSE	TRUE	
4	3	Bag	Perimeter	92.01	900	TRUE	FALSE	
5	4	Cure_Autoclave	None	0	3000	TRUE	FALSE	
6	5	Cure_Oven	None	0	2510	FALSE	TRUE	
7	6	NDI	Area	52	0	TRUE	TRUE	
8								
9								
10								
11								

Figure C.3, Design Table of pseudo process steps that could not properly be utilized by CATIA V5.

The limitations of CATIA V5 macros meant that looping through all rows of an arbitrary Design Table was impossible since there was no way of automatically determining the number of rows. It seemed that the desired functionality could have been achieved if the system included a few more options for interfacing to the Design Table. Apparently, this type of activity was not considered when CATIA V5 was designed.

C.4 Conclusion

Table C.1 summarizes the fundamental steps that were attempted to achieve a level of flexibility equal to that of Process Link within the CATIA V5 environment.

Step	Possible
Calculate area/perimeter values automatically from models	Yes
Read in external XL spreadsheet of cost equation	Yes
Evaluate cost for given processes	Yes
Automatically loop through, evaluating all processes	No
User-defined rules for process switching logic	Probably

Table C.1, Steps attempted for modeling custom manufacturing scenarios in CATIA V5.

Although only one row in this table is 'No', this results from quite significant limitations to how CATIA V5 interacts with the semi-external Design Tables. It can only handle data that relates directly to its in-built features types. Hence, it is impossible to implement general rules to manipulate data of other types (e.g. process steps equations, manufacturing task breakdowns, etc.).

Essentially, the types of activities investigated with the Process Link software prototype cannot be implemented without resorting to complex software that interacts with the CATIA V5 Application Programming Interface.

The principal benefit of CATIA V5 is the flexible and intuitive way to define constraints and relationships between parameters that influence a CAD model. To deal with composite manufacturing, the add-on modules discussed in Section C.2 would be required however these probably also lack flexibility for implementing non-standard data and algorithms. The main limitation is that all the feature types are in-built and the scripting programs and rules that are user-defined can only interact with the features in a limited number of ‘hard-wired’ ways.