An Investigation of Solid Modeling Practices in Industry

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Abstract

Parametric solid models provide a quick way of constructing parts that can be easily modified and redesigned for reuse in a variety of downstream applications. However, the method used to create the model has a significant impact on the level of usability. This research uses a combination of interviews, company standards evaluation, and model analysis to determine current industry practices for the creation of solid models. The focus is on creation of single part models. The paper includes a summary of differences in modeling methods based on designer preferences and software functionality.

Introduction

The goal of this study is to identify industry practices in solid modeling. A comparison of modeling practices used by experienced designers may be used to identify strategies that will improve or increase the usability of the solid part models and reduce the need for remodeling.

Models are typically started with a primitive or a sketched feature. Primitives can represent the stock material from which the parts can be machined. Sketched features produce geometry by extruding or revolving a 2D profile. The first primitive or the first sketched feature is usually located relative to a global Cartesian coordinate system. Additional reference entities, such as datum planes, axes, coordinate systems and points, are often created for constraining purposes. Subsequent features can be of any type and are constrained by references to existing part geometry. Adding the cosmetic features (e.g. fillets, chamfers, and threads) is typically the final step in the modeling process.

Background

The majority of solid modeling research, by far, is in the area of Product Data Management (PDM). This includes mostly database management, concurrent engineering and intelligent CAD (eg. Bronsvoort and Jansen, 1993). A small area of research is based on designing for assembly, where tradeoffs are made between the number of assembled parts and the complexity of each part (eg. Kim, 1997; He, Kusiak, & Tseng 1998). Other work being done in the area of modeling is for the purpose of finite element analysis. This involves the simplification of models to improve the efficiency and accuracy of the FEA (eg. Mathews, 2002; Rolf, 1995). Little or no work can be found in the literature regarding part-modeling strategies.

The majority of work dealing directly with the creation of individual solid models is presented at annual user conferences organized by the software companies themselves (Naujuk, 2002). However, these presentations focus mainly on the functionality of the individual software package and not on the general practice of solid modeling itself. The research presented here investigates the current solid modeling methods used in industry and generalizes the practices that apply to all major solid-modeling programs.

Research Objectives

The Part-Modeling Process

For a part-modeling process to be efficient it must follow a structured methodology for solid modeling. The methods used to create solid models may differ greatly depending on the particular downstream applications, product type, software functionality, required documentation and designer preferences. Possible solid modeling methodologies include a fabrication approach where the choice of features is based on the actual manufacturing process, e.g. material removal. Another method includes focusing on the constructive building blocks or sculpting/shape manipulation, where models are created using the simplest geometries to make analysis easier. This paper will use a combination of interviews and part structure analysis to capture some of the preferred

List of Interviews				
Interview	**Job**	**Company**	**Software**	**Part Types**
1	Designer	A	Pro/Engineer	Machine Parts
2	Designer	B	Pro/Engineer	Machine Parts
3	Designer	C	Unigraphics	Machine Parts
4	Designer	D	SDRC IDEAs	Mach./Molded Plastic Parts
5	Designer	D	Unigraphics	Car Audio Parts
6	Designer	E	AutoDesk Inventor	Computer Cases
7	Designer	F	Catia	Tooling
8	Standards	G	Pro/Engineer	-

Table 1

methods or best practices for the creation of solid models used in industry. A similar technique was used by Flemming, Bhavnani, & John (1997) in an investigation of practices used for 2D CAD (computer-based drafting).

Interviews of Model Designers and CAD Staff

The majority of the interviews were conducted with experienced mechanical designers, both engineers and CAD designers. All of these people used a solid modeling program on a daily basis. These individuals employed a variety of solid modeling programs. The level of experience ranged from those who started their design careers on drafting boards to those who started solid modeling right out of college. The parts being designed by this group included a combination of machine parts and product parts. Table 1 lists the interviews with company, jobs, software packages and type of parts modeled.

Study of Company Standards

Requests were made of the companies interviewed for any company standards regarding solid modeling. These standards only existed at the larger corporations, and company policies prohibited them from providing copies of the official documents. We typically obtained copies of the presentations used to present the standards to the designers. These documents demonstrated that the standards focused mainly on Product Data Management (PDM) and not on solid modeling best practices. However, some interesting results came from those standards that were collected, and are presented here.

Evaluation of Existing Parts

Files of designed parts were obtained from most of the designers interviewed. The standard practices used by that designer were extracted from these parts. The parts were evaluated using the native program to determine the particular method used to create the solid model. The model tree of each part was evaluated to determine the order of feature creation and the choices made for the constraint of each feature. Any specific profiles created by the user were studied to determine the design intent suggested by the specific parameters chosen. Finally, the results from the models were compared to the 2D drawings for similarities and differences in dimensioning schemes and associativity with the solid model.

Evaluation of a Common Part

Due to the wide variety of parts received for evaluation, a direct comparison of modeling styles proved difficult. The decision was made to have a number of designers each create a model of the same part. The part was chosen from the group of designs received. One part received from Company B seemed to use the widest variety of geometry and more choices for constraints were available to the designer. An orthographic drawing for the part was created from scratch, shown in Figure 1, using the same major features, eliminating any details referring to proprietary information. The simplified geometry and randomly chosen integer dimensions provided similar information without requiring a prohibitive amount of time from each designer. The drawing provided only the critical dimensions; all other geometry was to be scaled. The designers were asked to model the common part and return the native files to the research team. The part files were then studied to determine the modeling methods used by the designers.

Discussion

Interviews

The results of the interviews with the

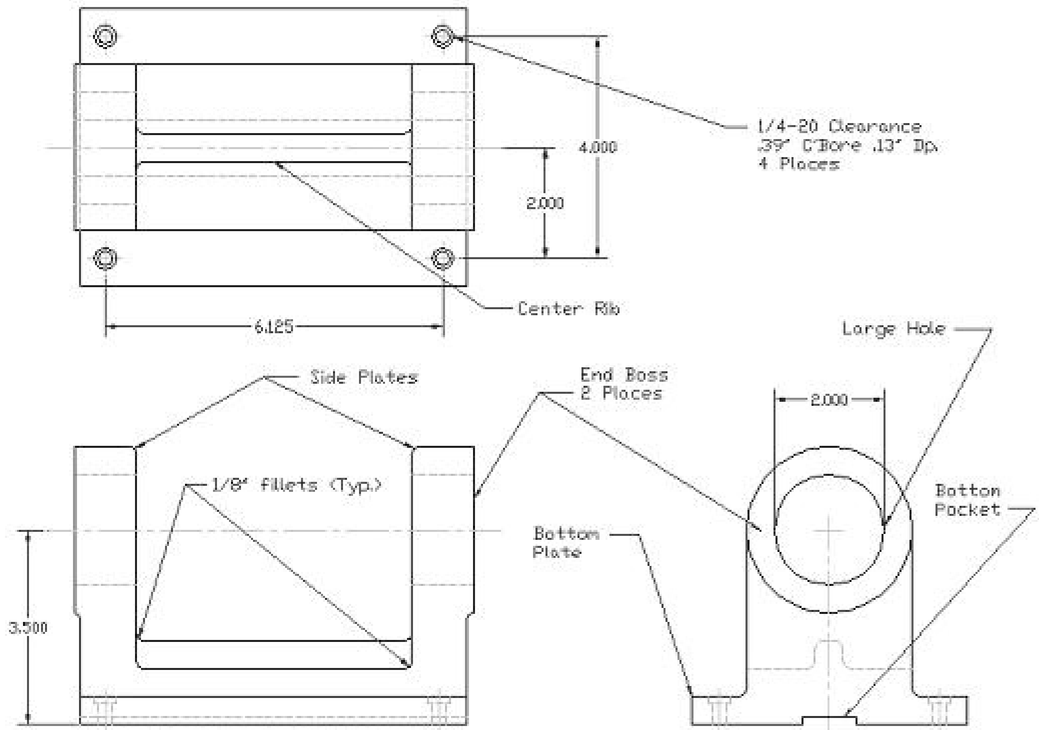

Figure 1 Drawing of Common Part

designers proved the most fruitful. A number of similarities were found and some unexpected differences. Some of these differences appeared to be the result of differences in software functionality. However, some differences in modeling strategies or methods were also found. The main similarities came in the form of general rules of thumb.

Designer Similarities

The most important similarity, also the most general, involves planning. Everyone agreed on the importance of doing some planning before any geometry is created. The purpose of planning is to determine the critical dimensions and references to provide for easy modification. Proper references allow for capturing the designer's intent, and facilitating future redesigns. Interestingly, the designers who had started on drafting boards recommended the part should be sketched on paper before modeling, but some others used the modeling program as a planning tool to quickly sketch out ideas. Upon completion of planning, it was not uncommon to start the final model from scratch. The process of designing versus strictly modeling produced different results. Designing is a dynamic process that usually changes direction a number of times before completion, creating a disorganized model. The model creation, after most of the design decisions are made, is quick and produces the cleanest model. However, this can never be done completely, and rarely does a designer look back at a complicated model and not wish something had been done differently.

Of those interviewed the consensus when selecting references was to define features with the same constraints and dimensions as those used to generate the corresponding 2D drawing. The selection of dimensions contained a mixture of goals. One common method is to design parts in a way that is consistent with the manufacturing process. The second, but equally important, is to prepare for the features and dimensions that are most likely to be modified. References and parameters are then selected to represent these dimensions as simply and logically as possible.

Parts are typically designed to mate with other parts. When parts are designed in an assembly mode, the parts use each other as references to ensure that they fit together. Parts that reference other parts are said to have external references.

Linking parts is a powerful tool; it facilitates the design of parts and reduces the chances of making errors while manually transferring data between parts. However, it was generally agreed that the links between individual parts should not be maintained beyond the initial design process. Once the model is finalized, the part should not change accidentally due to a modification of some other part. The software companies supply this functionality to allow for a continually self-updating model. However, most designers feel that these links are too difficult to maintain on a continuous level. External references are still used to help define the datums and matching geometry. However, these links are typically removed prior to finalizing the model, requiring some level of redefining.

Designer Dissimilarities

Some interesting differences were noted. Designers expressed different opinions about the use of sketches. Some expressed a preference to extrude or revolve complicated sketches in order to reduce the overall number of features that comprise a model. Other designers preferred to keep the sketches relatively simple and use additional features to create the complicated geometry. The latter group felt that simpler sketches and features produced a more manageable model. The software programs contain numerous tools, such as model trees, to manage features and how they relate to each other. Yet, complicated sketches do allow for easier relations to be made between parameters. The former group felt that the sketches communicate the majority of the design intent, and they strived to keep the model tree as simple as possible. These differences may be only a matter of different personal styles, but the differences are significant.

Differences also existed on the use of chamfers and rounds. Most users suggested chamfers and fillets should be applied towards the end of the model. A few felt this could cause errors with previously generated features. These problems are most likely a function of the software's method of handling these features rather than the method. Pro/Engineer treats chamfers, rounds and drafts as separate features that do not affect references of the existing geometry. Unigraphics, on the other hand, operates differently. Chamfers, rounds and drafts modify the identities of the surfaces they affect causing errors for other features referenced to that surface. For this reason, users of Unigraphics generally recommended that all geometry be referenced from separately created datum planes instead of from the surfaces of existing features. Chamfers, rounds and drafts are generally cosmetic and have minor importance to the main functionality of the part. It stands to reason that they should be added at the end of a model wherever possible.

Company Standards
Standards Focus on Documentation

The lack of official company standards suggests that most of the companies are interested, but have only begun to think about it. A few companies have been working on a set of standards but most of those were still in the rough draft stage or were created in house and not yet made official by the company. Due to the unofficial nature of these standards and due to proprietary issues, most companies were unwilling to show what they had produced so far. The standards found were not specific enough to solid modeling methods and generally covered more bookkeeping methods, such as colors, layers and file-naming conventions.

Company G Standards

Company G was unwilling to show us any official documentation, but instead provided the presentations used to explain the standards to the designers. The practices covered in these presentations focused mainly on file usage, program and system performance. Their largest concern seems to be concurrent engineering and database management, but the standards do list general practices and common problems. The standards relating directly to modeling involve the use of features. Unigraphics allows for the use of primitives, as well as predefined features and profile-defined features. Company G recommends the use of features wherever possible. Mixing primitives and features can cause dimensioning problems when creating drawings. Recommended practices include:

- Assembly cuts should be avoided especially cuts that remove features.
- Holes should be generated with the Hole tool instead of as a cut.

- Hole patterns should be used wherever possible.
- Excessive use of model cuts should be avoided.
- Simple profiles are preferred over complicated sketches.
- Dimensions should be driven by the model, not applied on the drawing as unassociated markups.

Company C Standards

Company C has produced a three-page document describing standards encompassing all CAD work. About one page of this covers the actual creation of solid models. The majority of this document covers bookkeeping practices, such as file management, format preferences, and preferences for units, start parts, layers and drafting settings. The section concerning solid modeling contained several general practices, some relating directly to Unigraphics. These include:

- All parts should be fully constrained on a coordinate system.
- Tolerance settings and body check should be used to ensure the geometry has no discontinuities.
- Final parts should be subtracted from a solid to check for cavity creation.
- Feature arrays should be used whenever creating a pattern of features.
- The copying or splitting of solids should be avoided.
- All features should be placed with positioning dimensions.
- Face offsets should be avoided.
- Symmetrical objects should be mirrored.
- The final part should not contain any suppressed objects.

The majority of these recommendations are specific to the functionality of Unigraphics. Unigraphics relabels surfaces when they are modified which causes linking problems with existing features that are referenced to these modified surfaces. The copying, splitting and offsetting of features is not a problem in other software packages, such as Pro/Engineer and Solidworks. However, fully constraining features, feature arrays and mirroring features seem consistent with most modeling practices.

Company D Standards

Company D has produced the most detailed documents regarding modeling strategies, including graphical examples. All of these standards are collected in an intranet website to provide easy access to the specific aspect required. The three main sections of this site are Part and Assembly File Structures, Standard Component Modeling and Non-Standard Component modeling. The latter two have graphically demonstrated examples. This website includes a list of general modeling practices with explanations of why these practices are superior. These explanations seem to be a key component in getting "buy in" from designers who have been doing it "their way" for a long time. These standards were created by the employees and have not been company mandated.

The Modeling Guide, contained in the website, discusses the advantages and disadvantages to the use of sketched features versus predefined features. Sketches are very powerful for creating complicated profiles and can easily convey design intent. However, sketches and their datums tend to clutter models and require more layers to maintain good file structure. Form features are a quick and easy way to generate simple geometry and can be easier to edit than some sketched features. However, form features have to be located explicitly to the adjoining solid. These connections can make it difficult to change their positions later. Primitives may be used to create the starting feature of a part but their use is not recommended beyond that. Other areas identified for improvement include the use of spreadsheets to control and edit expressions and the idea of creating and adding user-defined features.

In addition, Company D has created a number of generic part files representing some of the common geometry used in their applications. These parts, called Seed Parts, can be easily modified for any application, and are maintained in a central location so they can be modified to take advantage of any future improvement in modeling strategies. Company D has recorded significant reductions in modeling time for standard components when the appropriate Seed Part is used. The time to model a particular part was reduced from 16 hours to 4 hours and the resulting part was significantly easier to modify than the previously created models. Some employees felt that the Seed Parts provided more educational value than the instructional website itself.

Company D is clearly ahead of the other companies interviewed. Most companies are just beginning to consider standards. Company B has created an outline but is only beginning to write the document. Company F has a rough draft in process but was unable to show any of it. However, they did mention a central modeling lab where designers can be trained and the latest best practices demonstrated. Clearly, solid modeling best practices is still a young topic in industry. For true modeling consistency, company standards should generally require a more detailed description of modeling practices and contain demonstrations of case studies.

Existing Parts Wide Range of Results

Most of the people were reluctant to provide real parts to analyze. However, the parts received varied in type and shape, due to the fact that different industries were included in this study. From five companies, three designers donated parts. A total of nine parts were received; four were modeled using Pro/Engineer and five were modeled using Unigraphics. The results demonstrated a number of modeling styles. However, this variation might be considered a function of the various types of parts. In a number of cases, complicated profiles were used to create the initial geometry, and at the same time, it was also common to start with a simple shape and then add various profile cuts to produce the complex geometry. Finally, except for only a few departures, the critical features used to dimension the drawing were the surfaces used as the constraints in creating the model.

Part History's Effect on Design Intent

We were able to check the history of each model. The original expectations were that the older parts with a complicated history contained less intuitive design intent. However, no data were found to support this hypothesis. Unfortunately, most of the parts did not have a particularly complicated history. Most of the entries into the history tree represented various revisions of the start part. All but two of the parts had been created and modified by the same designer. The two models with two or more designers contributing to them were two of the better-designed models. This was not a large enough sample size to make a conclusion.

The Common Part

The focus of this trial was on the order and method use for feature creation. To determine the usability of each model, a number of changes were performed and evaluated for ease of modification. The greater number of dimensions changed or features redefined during each modification lowered the usability of each model. The two models evaluated varied greatly in style. The number of responses received was limited due to time constraints. Only two were readable with available software.

The first response, which was created in AutoDesk Inventor 7, had most of the geometry created with the extrusion of the first sketch. The second sketch was used to cut out the center section creating the two side supports and the center rib. Two datum planes, offset from the center plane, were used to create the boss on each end and the sketches were based off of the existing geometry. A third datum plane, based on the top surface of the bottom plate, was used to sketch and extrude the four holes. The majority of effort was put into the initial two sketches, a very efficient method.

The sketch to create the first feature, shown in Figure 2, was very intuitive and allowed for easy modification of the most of the dimensions. During the evaluation changes were made one at a time to the height and diameter of the large hole, the length of the part, and the location of the counterbored holes. The height and diameter were easy to change within the first sketch. One problem appeared when the length was changed. The two end-bosses were linked to two independent workplanes and not to the exiting geometry. This caused the bosses to remain in position while the other geometry moved. When questioned, the designer explained that he prefers to sketch on planes rather than on geometry surfaces. Sketches require more computer time to regenerate and the geometry gets regenerated more often than the datum planes.

Another complication arose when modifying the location of the counterbored holes. An extra

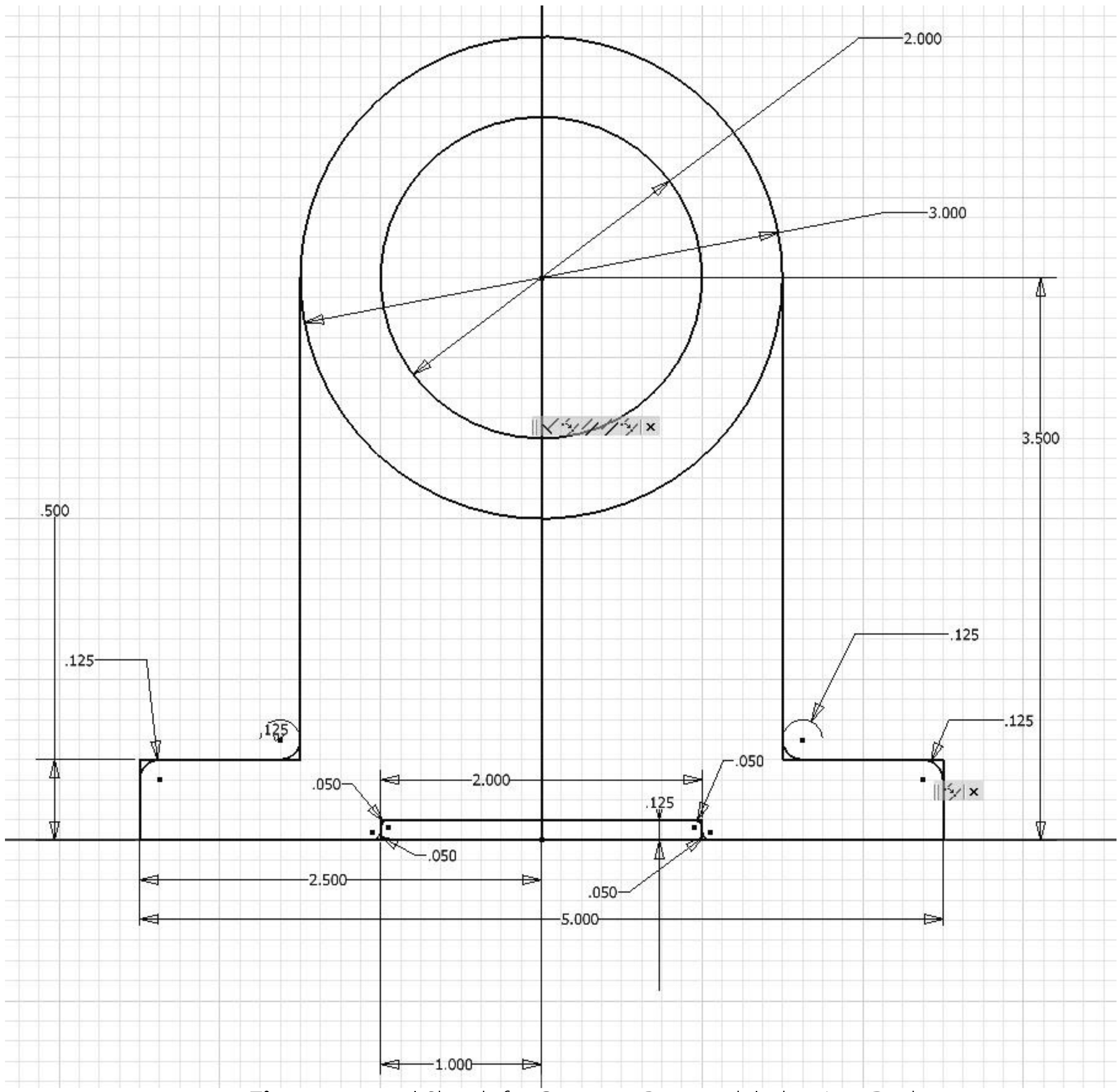

Figure 2. Initial Sketch for Common Part modeled in AutoDesk7

plane was created by the designer, so a problem would occur if the thickness had to be changed. The holes would stay with the plane and not move with the geometry. In addition, the Hole command wasn't used. Two sketches were used, one for the through hole and one for the counterbore. Changing the location of the hole would require the modification of two separate sketches in order to move the holes and since the patterning tool wasn't used each hole would need to be moved separately. The creation of the main geometry with two sketches worked well, but the creation of the secondary geometry (e.g. end bosses, holes) seemed to follow an "easy to model but hard to modify" methodology.

The second part based on Figure 1 was created by a different designer in Unigraphics, and contained more setup planning. It was built from the ground up. The order of geometry creation included the bottom two rectangular plates, the center rib, the two side plates, the counterbored holes, the two end-bosses and the large hole was last. The large hole was kept on the axis of the side blends by the use of a mathematical relation. This relation initially made modifying the height of the hole difficult, until the table containing all the parameters was studied to understand it better. The counterbored holes were made with the Hole and Patterning tools, making them easy to modify. The length was also modified without problems. The largest problem was that the heights of the side plates, clearly a critical dimension, were referenced from the top of the first plate and not the bottom. Using this dimension would be difficult when modifying the part. Overall, this part

Summary of Results					
Company	**CAD System**	**Fully Constrained**	**Primitives**	**Sketches**	**Fillets**
A	Pro/Engineer	Yes	No	Complicated	End
B	Pro/Engineer	Yes	No	Simple	End
C	Unigraphics	Usually	Yes	Simple	End
D	IDEAs	Yes	No	Complicated	During
E	AutoDesk Inventor	Yes	No	Complicated	End
F	Catia	Usually	No	Complicated	End
G	Pro/Engineer	Yes	No	Simple	End
H	SolidWorks	Usually	No	N/A	End
I	Pro/Engineer	Yes	No	Simple	End

Table 2

might have taken a little longer to create but proved easier to modify.

In comparison, the second model would seem superior. However, with a few modifications, the first would prove to be faster to create and easier to modify. The sketch used to create the main geometry was easier to understand and focused more on the critical dimensions. The recommended changes to the first method would include sketching the end-bosses directly onto the model surface and using the standard Hole and Patterning tools to create the counterbored holes. Essentially, this would be a combination of the two styles discussed here.

Conclusions

Consistent General Rules

Upon final consideration of all of the interviews and sample parts, the solid modeling methods used by various designers had much more in common than originally expected. The results are summarized in Table 2. The main list of common practices is as follows:

• All parts and features should be fully constrained. Despite some software ability to provide it, partially constrained geometry is generally not recommended.

• Simple or sketched features should be used wherever possible. Primitives are rarely used and, even then, only as the initial geometry of a part. Features provide more options for constraining and parameterization. Mixing features and primitives can cause different types of problems, depending on the particular software.

• Cosmetic features, such as fillets, rounds and chamfers, should be applied towards the end of the model and not have dependents. Suppressing these features to create a simplified representation is often required for downstream applications, such as FEA and large assemblies.

A wide variety of industries were sampled here. The parts designed in the industries sampled included machine parts, small molded/machined plastic parts, tooling and computer cabinets. At least six different modeling programs are being used throughout these industries. The differences in style may be a function of the type of part being modeled, as well as the software used.

Differences Between Designers

A few inconsistencies exist that seem to be due to personal preferences. The desire to use one complicated sketch versus a number of simpler sketches tends to vary from designer to designer. Some feel that the sketch provides the best method of communicating design intent. Creating all of the parameters in a feature eases the use of algebraic relations to control the geometry. The goal of using a complicated sketch is to keep the model tree as simple as possible. Others feel that simpler sketches are easier to manage and prefer to use the model tree to display the relationships among the different features.

Many designers also prefer to design using a material removal or machining methodology. This generally involves starting with the simple initial shape and using cuts to produce the desired parts. This method easily lends itself to designing for manufacture. It also encourages the use of the model dimensions as the drawing dimensions. Others feel that the modeling constraints and drawing dimensions are rarely the same and that the modeling parameters often need to change based on the downstream applications, e.g. NC, FEA.

Variations Related to Software

Some of the differences in modeling strategies can be attributed to the particular software's functionality. Occasionally, it was suggested that fillets and chamfers should be added throughout the process instead of at the end. The reason is that these features often cause errors if other geometry is referenced off of the surfaces affected by that chamfer or fillet. These designers also preferred to reference all features off of datum planes and not off of existing surfaces. This problem is a software issue with how Unigraphics identifies each surface and then re-labels it after being modified, causing problems with anything dependent on it. This is not a problem in a number of the other solid modeling programs available.

Not all designers inherit the model dimensions into the drawing. The dimensions in the inherited form are often too complicated to be usable. Designers using Pro/Engineer believe this to be a good practice and generally design this way. This suggests that inheriting dimensions is possible with the proper software functionality, although the type and complexity of part designed may affect the practice of inheriting dimensions.

Separate Designing from Modeling

The quality of a model is highly dependent on the amount of preplanning allowed. The situation producing the optimum results involves completely designing the part prior to the start of the modeling. Reproducing an existing design as a solid model can take, in many cases, less than an hour, whereas the original design may have taken days. Designing a part is a highly dynamic process and usually produces a less than perfect model. Decisions often change the model beyond the designer's ability to redefine the model in order to fully compensate for the new design intent.

Use CAD to Plan, Design and Model

Experienced designers recommend that the part be sketched out on paper before any geometry is created in the modeling program. This, however, goes back to the Flemming et al's(1997) T-Square metaphor where traditional methods are maintained for comfort rather than efficiency. Feature-based, parametric solid modeling can be an extremely powerful design tool when used properly. The designer needs to utilize these programs to plan and design the part. The easy addition and subtraction of features and the quick variations allowed by a parametric modeler can be used to plan the part and investigate as many design variations as needed. However, for the sake of the future usability of these models, the final model should be kept separate from the numerous twists and turns to the design process. In other words, the model should be fully redefined to reflect the final design intent or, if necessary, recreated from scratch. This will often require extra time, but will produce a better design and a more robust solid model.

Future Work

This article notes some interesting differences in modeling strategy. The next step would be to evaluate the efficiency of each strategy. But first, the harder task of defining and measuring efficiency in a meaningful way is required. The clearest difference between designers is the use of one complex sketch compared with the use of a group of simpler shapes. Therefore, the use of sketches should be the first subject of an efficiency study.

This article compared the results of a several designers modeling the same part, and produced some interesting results. However, a more complete trial should include different types of parts and separate out the differences between industries, software and strategies. We hypothesized that many differences in the results were due to the functionality of each software package, but there are also differences attributed to designer preferences and company practices. We are reminded of the "shoot-outs" that were often organized at conferences to demonstrate the capabilities of software packages in the early days of CAD. Similar studies could be organized to compare solid modeling software.

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