Abstract

Solid modelers provide a variety of tools that are useful in design, analysis and manufacturing of diverse components. One of the limitations of solid modelers is: they provide only common tools, which are useful mainly for general operations and not for specific ones. There are many design and application related problems, which can be solved only when some external functionalities, called plug-ins, are added to the modelers. In this project, some useful plug-ins have been identified on the basis of literature review, study of solid modeling packages and industrial survey. Based on the identifications, prototype of a plug-in, namely TopoDiff, has been developed. TopoDiff generates the signature of a model, which is helpful in comparing it with the similar models. This in turn saves a lot of time and efforts in the downstream processing such as analysis and manufacturing of the model.

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1. Introduction

1.1 Solid Modeling

Solid modeling has become the primary CAD technology for mechanical design. One of its main advantages is that the technology accounts for the connectivity of the various geometries (called topology) - something that a surface or wireframe system cannot do. Solid modeling can be used to create the bulk of components in a mechanical system, even thin-shelled parts such as those made of sheet metal or plastic. The advantages are many, including the ability to determine mass properties, perform structural analysis, or create STL files for rapid prototyping [1]. Moreover, solid modeling can perform some complex operations such as shelling that cannot be done with surfaces.

In the increasingly competitive design and manufacturing environment, solid modeling packages are drawing a great attention from the customers. The more versatile a modeler is, the more demanding it is in the market. What we mean by versatility is, the modeler should be user friendly, it should take sufficiently small amount of time in performing a task and it should be able to interact with other modelers.

Currently, several solid modeling packages, for example, Cadkey, CATIA, I-DEAS, Ironcad, Mechanical Desktop, Pro Engineer, Solid Edge, Solid Works etc., are in use for modeling and analysis of 2D and 3D objects. Most of them are made for Windows platform, while some of them such as CATIA, Pro Engineer and I-DEAS can be used on Unix platform as well. These packages are useful both from the design and analysis point of view. For example, **CATIA** provides products for intuitive specification driven modeling for solid, hybrid and sheetmetal part design, assembly design and integrated drafting. It also offers products for easy to use, designer-oriented part and assembly stress and vibration analysis for early pre-validation. One can create, simulate, optimize, document, build, and test one's products all within a single electronic environment: **I-DEAS**. It allows users to continue to work with, or migrate from, other commercial software applications. This helps in protecting investment in older CAD/CAM data. With **Mechanical Desktop** software one can perform virtually all the 2D and 3D design, analysis, and manufacturing tasks required for production from a single environment, faster and less expensively than with most other systems. It provides with more than 800,000 reusable 2D and 3D standard parts, holes, features, and structural steel profiles, as well as intelligent part placement and engineering calculations. **Pro-Engineer** is the de facto standard for mechanical design automation based on a parametric, feature based, fully associative architecture that delivers a comprehensive suite of solutions for all areas of the development process, from a product's conceptual design and simulation through manufacturing. **Solid Edge** is one of the easiest solid modeling system to learn and use because it was developed specifically for mechanical design professionals. **SolidWorks** is the standard in 3D product design mechanical design software for Windows. Its unique combination of production-level power, ease-of-use and affordability is highly efficient. SolidWorks automates the process of capturing and modifying design intent, enabling the users to move through the design cycle in record time. SolidWorks offers user-interface improvements as well as significant new file translation, detailing, assembly design, part modeling, and built-in migration tools that ease the move from 2D to 3D [2].

1.2 Problem Definition and Objective

Problem Definition Though, solid modeling packages provide a great deal of functions and utilities, there are many important functions, which are missing in the packages. This problem can be solved in two ways. The first one is to revise the whole package and come up with a more sophisticated version of the package. The second solution is to provide some external functionalities to the package. First solution can be provided only by the developer of the modeler and hence is somewhat rigid. Though, the packages are revised from time to time, most of the changes are general and are not related to a specific problem. The reason behind this is, different users have different priorities and hence their problems also substantially differ from each other. Hence the package developer mainly concentrates on the general traits rather on the specific ones. To take care of the specific applications, one needs to provide some external devices or tools that can be helpful in performing desired operations. These external devices are called plug-ins and are widely used in solid modeling packages.

Objective The objective of the project is twofold. They are:

i. Identification of a few plug-ins for solid modeling packages

ii. Development and Implementation of one of these plug-ins

1.3 CAD/CAM Plug-Ins

A Plug-in is an additional feature, which is provided to a software or hardware to improve it's functionality. Plug-ins are extremely useful in those situations where we are more concerned with specific problems. In case of solid modeling, plug-ins are those external *tools*, which are added to the package to improve its functionality and help users in tackling specific problems. For example, one CAD/CAM user may be concerned only with modeling aspect while the other one may be dealing with other aspects such as data exchange etc. To be more specific, a CATIA user may want to export a file to ProE but at the same time he might not be having anything to do with the Solid Works or Solid Edge. In such cases, plug-in utilities are appropriate solutions to such problems.

. Plug-ins are normally provided by the users of the modeling packages and since they are not inherent parts of the modeler, same plug-in can be used for different modelers with or without moderate alteration. For example, a plug-in made for Solid Works can be used for Mechanical DeskTop with a little modification.

There are various kinds of plug-ins that can be used for the modeling packages. Some are geometry related, some are related to finding errors and some can deal with data exchange, among others. An example of geometry related plug-ins is one, which finds the difference between two models that differ with each other in terms of some particular features. On the other hand, error-finding plug-ins are used to detect any error relating to dimension or geometry in the part or assembly whereas data-exchange plug-ins are mainly used for import or export purposes. Sometimes, parts from one package need to be modified in the other packages. To handle such problems, an import/export plug-in can be used

1.4 Approach

The following approach was adopted to achieve the objective:

1.4.1 Study of Solid Modeling Packages

Since the project mainly deals with the solid modelers, a study of well-known modeling packages is necessary to have the understanding of the problems. To understand the different features of a modeler, it is best to perform 3D modeling (called Hands-on) on the modeler. Three packages, namely Solid Works, Solid-Edge and Mechanical DeskTop were used for this purpose. Also, documentations of two modeling packages, namely CATIA and I-DEAS, were studied. After studying the solid modeling packages, a comparison table was prepared to compare the strengths and weaknesses of these packages. The aim was to identify some tools, which were either absent in the modeler(s) or were relatively inefficient. These tools could be identified as plug-ins.

1.4.2 Industrial Survey

Industrial users of CAD/CAM softwares are well experienced with the packages and hence they can better tell the difficulties with the packages. Also, since they are associated with specific industries, they can point out application related problems as well. Having this purpose in mind, an industrial survey was conducted. The results of the survey were used in identifying some more plug-ins.

1.4.3 Identification of a Few Tools

Based on the results from study of CAD tools, literature survey and industrial survey, a few tools were identified, which can be developed as plug-ins.

1.4.4 Prototype Development

After identifying a few plug-ins, one was selected for the development purpose. The environment used for the development was Visual C++.

This project was done in collaboration with Geometric Software Solutions Co. Ltd. (Geometric). Geometric is a leading provider of software applications, component technology and development services to mechanical design and manufacturing markets worldwide [3].

1.5 Organization of the Report

The report is organized into five chapters. Chapter 2 summarizes the literature review done in design and application related fields of the solid modelers. Chapter 3 discusses the solid modeling packages and their functionalities. 2D tools have been mentioned and 3D tools have been discussed in detail in this chapter. Also, comparison table and industrial survey along with its results have been discussed. In the next chapter, TopoDiff and its relation to GeoDiff have been discussed. In the following chapter, features of TopoDiff have been put forward to. In the last chapter, we summarize the contribution, limitations and future work of this project.

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2. Literature Review

Design and application-oriented literature review was conducted to identify some plug-ins that can be used in solid modeling packages. Design-oriented plug-ins are useful in the geometry related problems whereas application-oriented plug-ins are used to solve process related problems. Application part is equally important because applications are specific to particular industries. Based on this literature survey, a few tools were identified, which could be developed as plug-ins.

2.1 Design Functions

Design automation is a broad term that can apply to almost any CAD tool. But for mechanical design, it has a real meaning that cuts to the core of what engineers are trying to do with CAD: create and complete design faster. Another benefit design automation tools bring to the modeling and drawing process is consistency and accuracy. [4]

Design add-ons usually come with many tools. Some developers offer these functions in one product; others provide separate modules. Design functions can generally be classified into one of these areas:

2.1.1 Part Libraries

Part libraries contain models of standard parts such as bolts, nuts, washers, etc. that can be simply inserted in the drawing or model. Typically, rather than a static chunk of geometry, these parts are parametric so we can easily customize them to suit our needs.

2.1.2 User Interface

User interface enhancements are meant to improve the way we interact with CAD systems, providing additional toolbars, dialog boxes, and other items to save time and effort.

2.1.3 Feature Libraries

Feature libraries make it easy to add features to our models that go beyond those found in the standard CAD system tool sets. These functions help us speed up geometry creation by providing features that we can use instead of having to create them ourselves. As with symbol libraries, parametric capabilities allow us to fine-tune these features to our needs and to access their properties for later revision.

2.1.4 Built-in Calculators

It allows doing the math for making design decisions, help to select or size particular components, or evaluate options.

2.1.5 Dimensioning

Dimensioning tends to be one of the most time consuming tasks in design and one that varies from company to company. Developers provide a plethora of tools to assist in creating dimensions, symbols and other annotation tasks.

2.1.6 Knowledge-based Design

This software creates rules and relationships that determine what components are to be used as well as the size of the components based on input from the users. These rules are then used to drive the CAD systems to automatically create models and drawings.

This overview provides a description of some of the design tools for various CAD packages.

2.2 Application in Industry

There are many industries, for example, mold and die industry, sheet metal industry, machine tool industry, rapid prototyping industry, shipbuilding industry, etc. where CAD/CAM packages are extensively used. These industries require special functionality for specific operations. The facilities provided by the prominent solid modelers are general and not specific to any particular operations. We will look at some of these industries in detail.

2.2.1 Automobile Industry

Assemblies are the main components used in automobile sector. Until only a few years ago, engineers designed parts individually and then pieced them together later in the development cycle to see if they fit properly and the product functioned as intended. Such a system was fine for small design teams working on simple problems. But larger groups often had problems designing complex assemblies as part of an overall product. A designer might change a component configuration and fail to let others know, or forget to make corresponding changes in other affected components. Considerable time was spent manually tracking part- designs, part-to-part interfaces, engineering changes, product specifications, test results, and other critical information to make sure individual part designs were in sync with one another. In assembly one needs to adopt a top-down design approach, where engineers start with system-level functions and specifications and gradually work their way down through various levels of detail until performance parameters are determined for individual parts. This contrasts with traditional bottom-up approach in which parts are first designed and then pieced together into an overall system, which then require extensive physical testing and mockups to determine if all of the components fit together and function as required.

The aim of assembly modelers is to provide logical structure for grouping and organizing parts into assemblies and subassemblies. Individual identification of parts and their associativity with other parts is important. Part information usually includes some type of alphanumeric designation tagged to the CAD model, which identifies the type of component, the particular configuration and the revision level of the design. Relationship data maintained by an assembly modeling system includes a wide range of information about the part and its association with others in the assembly. Mating conditions identify how the part is connected to others, as in a fixed position or by a rotational joint, for example. Operational parameters specify the distance the part can move with respect to mating parts, such as the number of degrees it can rotate or the linear distance a pin can move in and out of a hole. “Instancing” information identifies other places in the assembly where the same component is used, which is important for tracking common parts such as fasteners [4]. Data on fit, position, and orientation specifies exactly how parts are put together in the assembly with respect to one another, often important information on tolerances. We also require the capability to create parametric constraint relationships between parts and to measure size and dimension information from one part and apply it to another, which will allow user not to re-enter geometric data where parts interface. Once such a relationship is established, the user need only change one dimension, and the other related to it change automatically. Constraints can be static measurements, so that the height of one part always equals that of the one beside it.

2.2.2 Mold and Die Industry

Ten years ago, it typically took a month to prototype mold and another three months to complete mold design and manufacturing. Today, however, customers want prototype molds in less than one week and finished molds in four to six weeks. That's why mold making veterans seek out technology to stay competitive. One class of technical tools they can't do without is software that speeds mold design and manufacturing. Such programs enable users to refine CAD models and add machining geometry. Moreover, the software enhances NC programming and supports an array of machining operations to help reduce overall cycle time and improve quality [5].

Unlike general-purpose machining, in which manufacturing engineers work with solid models, mold makers typically work with surface models. In addition, mold makers deal with geometry often represented by doubly curved surfaces. Surface edges are rounded with fillets and blends, and flat surfaces require draft angles. The numerical control program to machine the molds often consist of dozen or even hundreds of tool paths that are run once. This differs from general-purpose machining in which the programs consist of one or a few tool paths that are run multiple times.

Roughing, semi-finishing and finishing operations are essential in mold making. The most common type of mold or die produced is for injection molding. When it comes to design plastic-injected molds, software vendors offer different perspectives on what functionality and features are important to the process. In part, the perspectives are shaped by the customers they serve and products they offer. It is important for users to have a complete set of modeling tools and options for generating basic designs. Also, a concurrent engineering environment-including software that allows design models to be changed while the mold geometry is still being designed- is critical. Also, for complex molds, large assembly capabilities and associativity features are musts. Model makers require powerful CAD functions to make part modifications as well as add machining geometry not typically included in CAD models. Solid models provide a well-defined starting place from which they can build the final molds.

Shrinkage adjustments can be programmed in as a single factor rather than trimming, extracting, or offsetting multiple entrances. Plus, solid modelers can help create other mold-associated geometry such as gates, sprues, runners, and ejector pinholes. Users also look for software that allows core and cavity surfaces to be separated automatically and that can automatically determine the natural parting line for shapes and use that to create split surfaces because plastic molds must have parting surfaces to be removed from the mold, software must also be able to create or modify the design on surface models. Moreover, the tool should allow for multi-surface, variable radius fillet to be created quickly and easily. In some instances, moldmakers rely on Electrical Discharge Machining (EDM) operations. EDM is appropriate for use in applications where detail and accuracy count. In EDM process, neither the tool nor the workpiece rotates, resulting in the ability to produce sharp corners. Electrodes often contain complex shapes, sharp angles and need to be fitted to a specific area. Therefore, electrode should be created using an existing solid part: by choosing the area and using a Boolean operation to remove the electrode tool from the part [6].

2.2.3 Shipbuilding Industry

Upgrading to a new 3D modeling approach has helped shipbuilding industry reduce the time and cost involved in building ships by providing enhanced design products to be used in construction. The new 3D approach allows designers to define where piping and equipment exist in the structure, so all piping, equipment foundations, and structure can be prebuilt in sections or subassemblies and brought together for assembly into a single unit. Previously piping and structural design was done separately; hence it took an enormous amount of time to calculate the intersection of piping and structural members. The basic idea is to use a 3D CAD program to create a single model that includes all parts of the ship, structure, piping, equipment foundations, etc. This would save substantial time in both design and construction [7].

Therefore, we see that various industries have their own design and manufacturing problems, which are not addressed by solid modelers. Here, plug-ins are extremely helpful and can be provided to the solid modelers to take care of specific problems. Based on the above study and other sources, following new tools can be considered as candidates for being developed as plug-ins:

2.3 New Tools

Based on the above studies, a few tools were identified as candidates for being developed as plug-ins. Different tools have different applications and a tool that is extremely helpful in one application may not be useful in some other applications.

1. **Check Dimensions**: This tool can be helpful in checking all the dimensions in an assembly so that components fit fully.
2. **Combining Solid modeling and Surface modeling**: The benefit of adding solid and surface modeling is that we have the best of both worlds. Normally, surface modeling is used to design freeform surfaces and solid modeling for designing simple, prismatic surfaces [8].
3. **Automatic hidden line removal**: Sometimes hidden lines pose a great deal of problems to the users. Hence automatic hidden line removal tool would be extremely helpful in that case.
4. **Shape- Relationship symbols**: There is a definite relationship between different shapes of a model. This relationship is defined by the constraints. The shape-relationship symbols tool can identify those constraints and highlight them.
5. **Specify, modify and delete relationships**: Between geometric entities such as holes, bosses and ribs, or even between different parts, we can specify, modify or delete relationships such as parent or child, etc. Parameters can be logical, numerical values as well as equations.
6. **Automatic generation of Bill of Material:** This tool is helpful when we need to have the information about the parts being used in the assembly.
7. **Automated multi-cavity and runner tools:** This tool can be used for preparing, modifying and analyzing single and multi-cavity molds complete with sprue, runners and gates.
8. **Automatic display and removal of grid lines:** Sometimes grid lines are unwanted from the point of view of modeling and at the same time sometimes they are required for better understanding of the model. This tool gives the facility to display or remove grid lines [9].

9. **Isolate selected geometry**: Sometimes the model is very complex and large (specially in case of assembly) and we want to work only on a small portion of the model. This tool can be used to select a small portion out of a larger one to simplify operations.
10. **Specify mating condition**: We can specify the distance parts can move with respect to each other or else we can specify the number of degree it can rotate in case of rotational joint.
11. **Instances**: This tool can be helpful in identifying other places in the assembly where the same component is used, which is important for tracking common parts such as fasteners.
12. **Measure size and dimension**: This tool is helpful in measuring size and dimensional information from one part and applying it to others, which helps to free users from re-entering data.
13. **Inter- part constraint**: When a part is modified, the related parts need to be updated. E.g. if diameter of shaft changes, the size of the holes it fits into should be updated as well. This tool is used to serve this purpose.
14. **Simplifying big assemblies**: This can be done by modeling commonly used parts only once and specify the multiple locations. So, we need to change only master part [10].
15. **Disappear internal features**: This tool can be used to make model look less complicated.
16. **Divide into zones**: This is useful when the user wants to work in specific areas, so that the model can be divided into cubes or blocks. One area can be modified without going into detail of others.
17. **Functional Subdivision**: This tool can be used when we require functional subdivision such as mechanical, hydraulic or electrical.
18. **Master drawing**: To be used as templates for creating drawings that conform to the standard.
19. **Locate pipes**: In ship industries, locating pipes is a complicated problem and specific tools are required for this purpose.

In the next chapter, we will study some well-known solid modeling packages in detail to understand their usefulness in design and manufacturing industries.

3. Study of Solid Modeling Packages

3.1 Introduction

Since the problem mainly deals with solid modelers, it is necessary to understand the functionalities of different solid modeling packages. This would help in acquiring the knowledge about the existing and missing tools in the solid modelers. Also, sometimes the same CAD tool is more efficient in one modeler than the other. Hence, a thorough understanding of well-known solid modeling packages and their strengths and weaknesses is required.

3.2 CAD Tools

There are numerous Computer-Aided Design tools present in each solid modeler. The efficiency of a modeler depends on the ease with which an average user can use these tools. Also, the usefulness of these tools varies from one package to other. Study of CAD tools is the first step towards identifying some plug-ins, which are not present in some of or all the solid modeling packages. CAD tools can broadly be divided into two domains, namely 2D Tools and 3 D Tools.

3.2.1 2D Tools

2D tools are used to create sketch on the sketch plane. To draw any sketch, we need to have some basic tools necessary for that sketch. Some important 2D tools are as follows:

1. Line
2. Circle by centre
3. Arc
4. Polyline
5. Rectangle
6. Spline
7. Horizontal line

8. Vertical line
9. Distance between
10. Parallel
11. Perpendicular
12. Tangent arc
13. Angle
14. Equal
15. Coincidence
16. Concentric

These tools are common tools, which are provided by almost every solid modeler. Most of the above names are self-explanatory e.g., Line is used to draw a line, circle for drawing circle and so on. Some of the above tools show the constraints, e.g. "Distance between" gives the distance between two entities, "Perpendicular" shows the perpendicularity between two lines, "Coincidence" shows whether two lines are coincident or not and so on.

Most of the 2D tools are inherent in the solid modeling packages and there is little need of providing them externally. They are very simple and basic tools and hence are present in almost all the well-known solid modeling packages.

3.2.2 3D Tools

3D Tools are extremely important from the modeling point of view. There are more than hundred of 3D tools, which are in-built in a modular. But, these tools vary in their functionality and effectiveness from one package to another. The same tool can be quite cumbersome to use in one package and very easy to handle in the other one. Also, speed is the other factor, which must be given considerable attention. Modelers vary from each other in terms of speed. Obviously, customers look for the fastest one. Following tree-chart represents the important 3D tools:

a) Sketch- Based Features

Sketch- Based Features are the ones, which are used to convert 2D profile into 3D features. For example, a circle, when given a depth, converts into a hole or cylinder.
Following are the important Sketch- Based Features.

i) Pad

Pad is the tool, which creates extruded surface. The limit on this surface can be of different types. A profile can either be extruded to the next feature it encounters, or it can be extruded up to the last feature, or, up to a particular plane or surface. Normally all the extrusion in the direction perpendicular to the base plane, but, there are a few modelers which have the facility to extrude in the direction not perpendicular to the base plane.

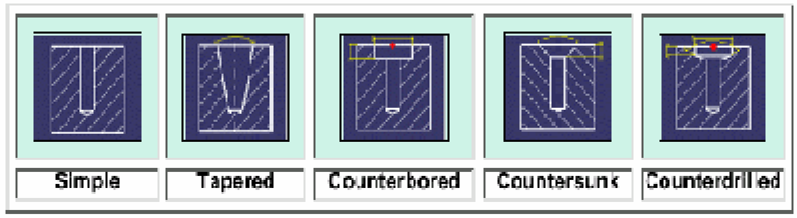
ii) Pocket

Pocket is similar to Pad except that it is used to remove material rather than adding it. Rests of the functions are exactly similar to those of Pad.

iii) Hole

Hole is a very crucial tool for creating different kind of holes as per the requirement. Holes are categorized broadly as simple, tapered, counterbored, countersunk, counterdrilled and threaded.

Simple holes require only diameter and depth. Tapered Holes require angle also, in addition to the above two parameters. Counter bored holes need two diameters and two depths, one each for counter and main hole. Counter sunk hole must be given angle at which the hole has to be sunk. Threaded holes need to be given pitch as the parameter.
Apart from the diameters and depth hole must be specified in terms of its extent. Hole can either be blind or through or can go up to next, up to plane or up to surface. The following figure illustrates different types and modules of holes.



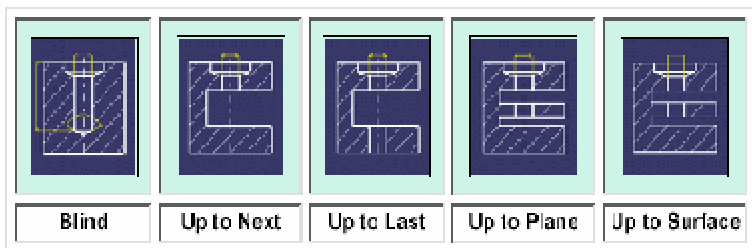


FIGURE 3.1 HOLES TYPES AND MODULES

iv) Shaft

Shaft is a revolved feature. To create a shaft, we need to have a profile and an axis about which the profile may be rotated. The angle of revolution can lie between zero degree to 360 degree.

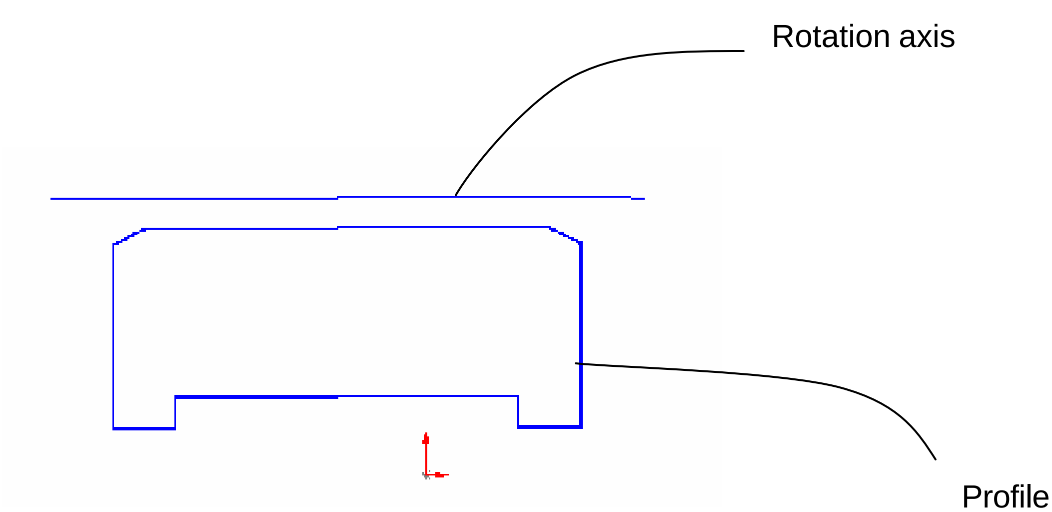


FIGURE 2.2 CREATING A SHAFT

v) Groove

Grooves are revolved features that retrieve materials from existing features. Here also the angle can be anything between zero degree to 360 degree.

vi) Rib

Sweeping a profile along a centre curve by creating material makes ribs.

vii) Slot

Slots are similar to rib but hare material is removed while sweeping the profile along a centre curve.

viii) Stiffener

Stiffner is an extruded portion that is created by giving a thickness to a profile and extending it in the direction perpendicular to the direction of the thickness. The extension can either be in one direction only or in both directions symmetrically.

ix) Loft

Loft features are created by sweeping one or more planar section curves along a computed or user defined spline.

b) Dress-Up Features

Dress-Up Features are those features, which are used to give a finishing touch to the parent features. Fillet, chamfer, draft, shell and thickness are some prominent dress-up features.

i) Fillet

A fillet is a curved face of a constant or variable radius, that is tangent to, and that joins, two surfaces. Together, these three surfaces form either an inside corner or an outside corner.
Fillets are one of the most important features, which are extensively used. There are several kind of fillets, viz. Edge fillet, Round corner fillet, Face-face fillet, Tritangent fillet, Variable radius fillet etc.

Edge fillets are smooth transitional surfaces between two faces. Round corner fillets are fillets whose ends have been rounded off. When there is no intersection between the faces or there are more than two sharp edges between the faces, Face-Face fillets are used.

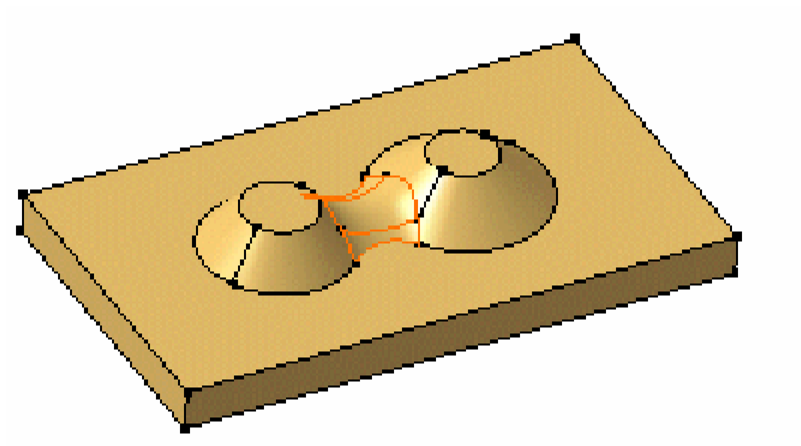


FIGURE 3.3 FACE-FACE FILLETS

Variable radius fillets are curved surfaces defined according to a variable radius. A variable radius corner means that at least two different constant radii are applied to two entire edges.
The creation of Tritangent fillet involves the removal of one of the three faces selected. The removed face acts as a fillet between the rest of the two faces.

ii) Chamfer

Chamfering consists in removing or adding a flat section from a selected edge to create a beveled surface between the two original faces common to that edge. By specifying the length and the angle, we can create a chamfer.

iii) Draft

Drafts are defined on molded parts to make them easier to remove from mold. To define a draft, we need to specify a pulling direction and the draft angle. Sometimes, it is wise to define parting element and neutral element while drafting the feature.

iv) Shell

Shelling a feature means emptying it while keeping a given thickness on its sides. Shelling may also consist in adding thickness to the outside.

v) Thickness

Thickness is used to provide extra thickness to a surface or plane. It can be used to remove material as well.

c) Edit

Undo, Redo, Cut, Paste, Search, Update, Delete etc. are the main editing tools. Undo cancels the previous operation. Redo redoes the previous operation. Cut and Paste are the usual routine functions. Update is used to update the existing model. In some softwares, the model is not updated without explicitly giving the update command. Search is used to search for some sub-parts in a part or assembly.

d*) Surface-Based Features*

Surface-Based features are obtained by applying commands to the surfaces or by using surfaces for modifying features of any type. Important Surface-Based features are:

i) Split

Split is used to split a body by means of a surface. The splitting surface can be selected by the user and the portion of the body to be removed or to be kept can be determined as well.

ii) Close Surface

This tool is used to close the surfaces that are open at one or both ends. A box with the open top can be easily converted into a closed box using this tool.

iii) Sew Surface

Sewing means joining together a surface and a body. Using this tool results in computing the intersection between the surface and the body while removing useless material.

iv) Thick Surface

Material can be added to a surface in either direction using this tool.

e) *Transformation Features*

Transformation features are obtained by applying commands on existing features. Following are the prominent transformation features:

i) Translation and Rotation

Translation and Rotation are the routine features used to translate and rotate the current body to the desired location. The translation distance and the rotation angle is provided by the user.

ii) Symmetry

Symmetry is used to create an exactly similar element using a point, line or plane as the reference element. The original element is no longer visible after using this command.

iii) Mirror

Mirroring is used to duplicate a body or feature using symmetry. There must exist a face or plane about which, the body can be mirrored. The original element is unchanged after performing the mirroring operation. If the original body is later on changed, the mirrored element gets automatically modified.

iv) Rectangular Pattern

Patterning is a very useful operation used to create multiple copy of a feature on the desired locations. The rectangular patterns are used to create copies in a rectangular matrix. There are three parameters, namely *Length*, *Spacing* and *Instances*, which can be used to create rectangular pattern. Using any two of these gives the pattern as required.

Spacing is the distance between two consecutive copies of the feature. Spacing in both directions has to be specified. Instances is the number of copies of the feature to be patterned. Length is the length of the body on which patterning is being performed.

The two directions of the matrix must be pre-specified.

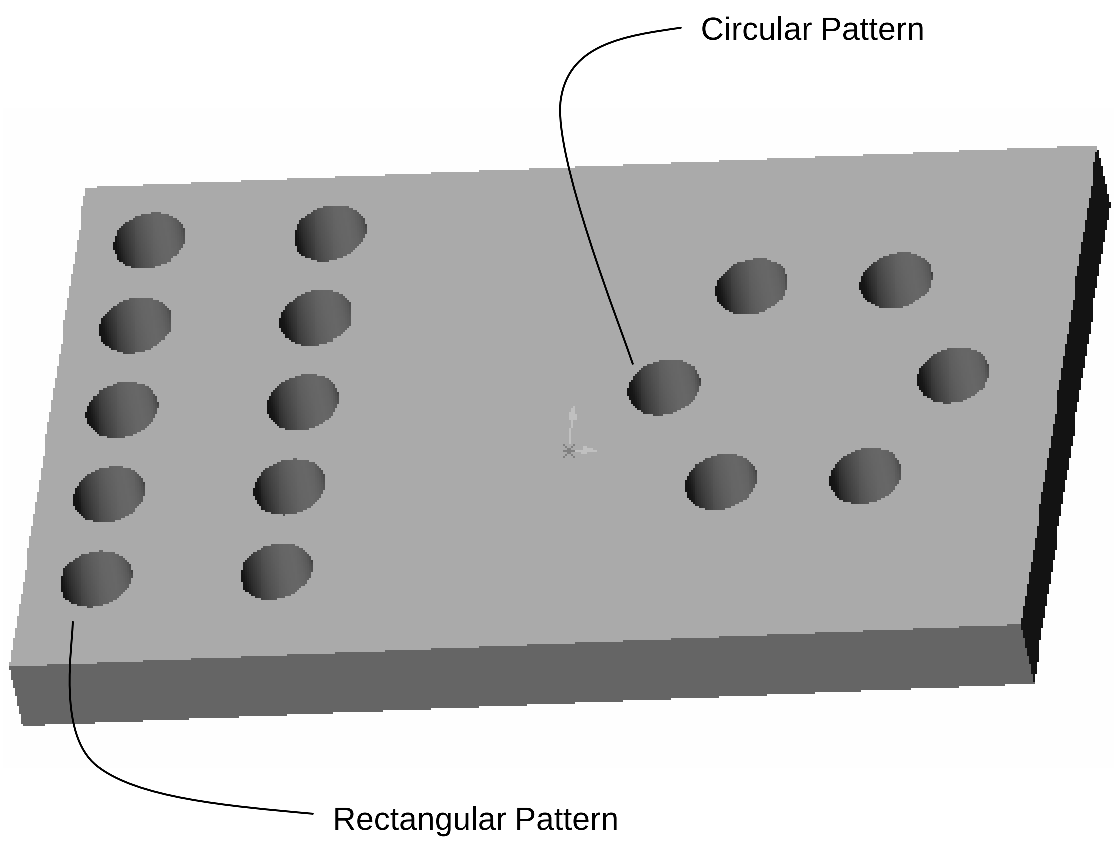


FIGURE 3.4 PATTERN

v) Circular Pattern

Circular patterns, often more convenient than rectangular patterns, are used to duplicate a feature in a circular manner. The rotation axis, about which, the patterning is to be done, must be consistent with the specifications. The parameters involved with the circular pattern are:

i) Instances and total angle,

ii) Instances and angular spacing,

iii) Angular spacing and total angle,

iv) Complete crown

Any of the above parameters completely and uniquely determines the pattern. Instances denotes the number of copies, total angle is the angle along which the patterning has to be done, angular spacing is the angular distance between two consecutive copies. Crowning is a special feature, which duplicates a whole circle at a specified distance [11].

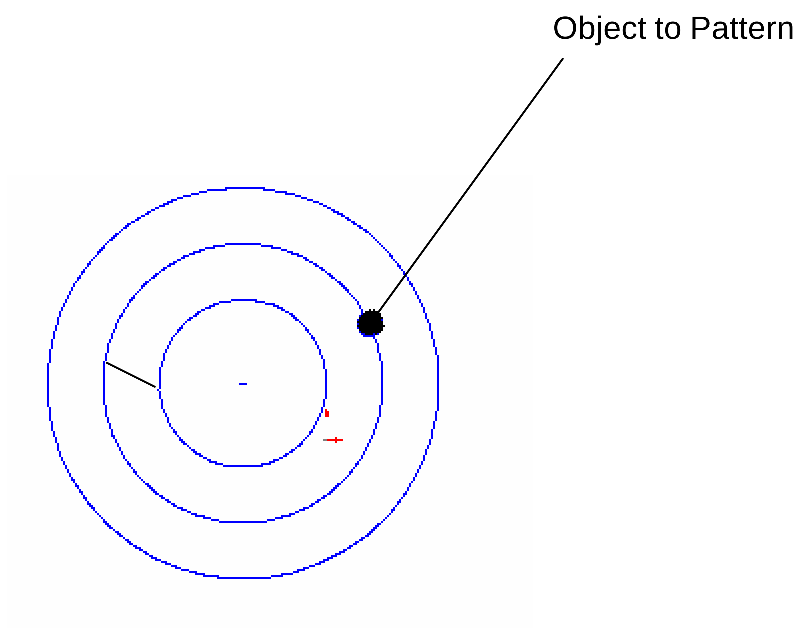


FIGURE 3.5 CROWN WITH DISTANCE GIVEN

vi) Scaling

Scaling a body means resizing it to the specified dimension. Scaling can be done with respect to a point or a face or a plane.

e) Boolean Operations

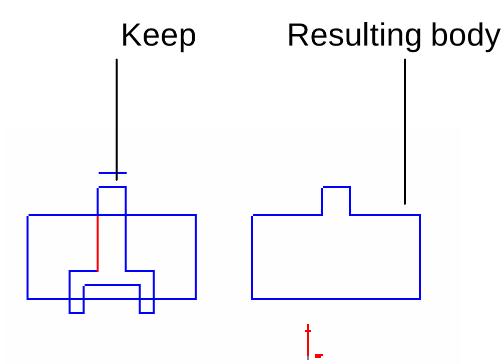
To design a part, we need to create features within the same Part body but we also need to insert additional bodies, which are combined together in various ways to create material. Once our bodies are well defined, we can assemble them performing an assembly or compute their possible intersections. We can also add or remove bodies from other bodies or even use the trim capability, which combines addition and removal of material. The different ways of associating bodies to form a part are: Assembling Bodies, Intersecting Bodies, Adding Bodies, Removing Bodies, Trimming Bodies

Assembling is the operation integrating the part specifications. The bodies are assembled depending on the selection of the part body. The Intersection operation finds the material shared by the two intersecting bodies. Adding and Removing operations are used to add or remove a body to or from a part.

f) Trimming

Trimming operation is performed to determine which element to keep and which to remove while dealing with two bodies. Concretely speaking, we need to select the two bodies of

Rule 1



Rule 2

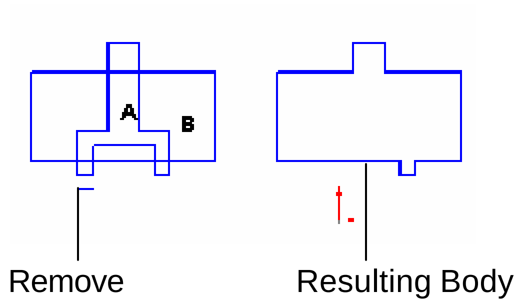


FIGURE 3.6 TWO RULES FOR TRIMMING

interest and specify the faces we want to keep or remove [12]. There are two rules guiding trimming operation, which are illustrated by the figure.

g) Measure

Measure is a very useful tool, which tells about the distance and properties associated with the features and parts. There are two kind of measuring operations.

i) Measure Between

This tool determines the distance and angle between two geometrical entities. The two entities could be vertices, edges, surfaces or entire products. Out of all measuring elements, the one measuring minimum distance is of extreme importance for this is often used to put some constraints on the features.

ii) Measure Elements

This tool is used to measure the properties associated with a selected element. The elements and their important properties are:

Element	Properties
Point	co-ordinate
Edge	Length, Angle, Radius, Start Point, End Point, Centre Point
Surface	Area, Centre of Gravity
Volume	Volume, Area, Centre of Gravity.

Area, Volume and Centre of Gravity are properties, most commonly used.

h) Updating

After performing any new operation on a feature, the feature needs to be updated. This updating can be of two types:

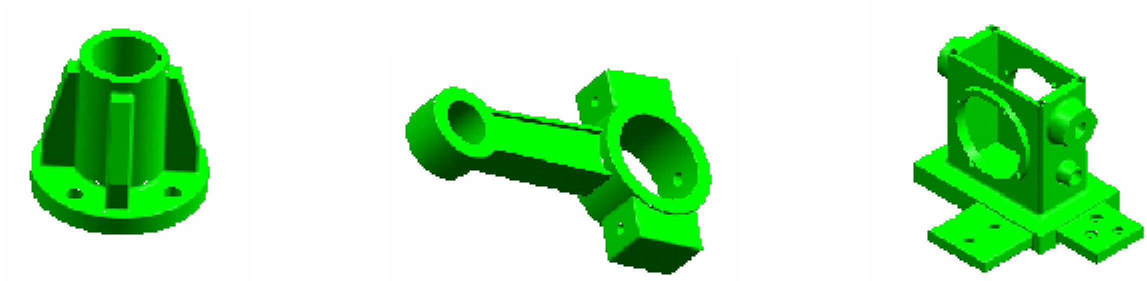
a) Manual Updating,
b) Automatic Updating

Most of the modelers have in-built automatic updating but some modelers do provide manual updating (that is, by clicking or giving a command) leaving on the users to update as and when they want.

3.3 Comparison Table

After studying the CAD tools, a comparison table was prepared featuring the important aspects of various solid-modeling packages. The table contained the information about diverse tools present in the solid modeling packages. Before preparing the comparison table, hands-on was performed on three solid modelers, namely Solid Works, Solid-Edge and Mechanical DeskTop, in order to better understand these packages. Three models were, namely *Bracket*, *Connecting Rod* and *Block*, were chosen for this purpose.

The first two parts were relatively simple and the third one was a complex part. The Bracket had 687-kb memory, the Connecting Rod had 766-kb memory and the Block had 890-kb memory. The hands-on was performed on a Window platform with 128 MB RAM. All the models in three modeling packages were created on the same computer, which eliminated the possibility of effect of speed on the modeling.



a. b. c.

FIGURE 3.1: A) BRACKET, B) CONNECTING ROD, C) BLOCK

After studying the various packages and performing hands-on, a comparison table was prepared featuring the presence and absence of diverse tools in the packages. Following table presents the content of the table:

3.4 Industrial Survey

A number of companies offer CAD systems. The selection of a specific CAD system by an industry is governed by the following factors: the intended end use, the functionality offered by a system in support of such end uses, and the cost of acquisition and training associated with the CAD system. While CAD systems continue to evolve, there are still many improvements, related to functionality and productivity desired by engineers. Keeping this in view, an industrial survey was conducted in Geometric Software Solutions Co. Ltd., Pune in November 2000. More than 50 participants from various parts of the country, mainly belonging to the leading companies who use CAD/CAM packages, came to participate in a seminar-cum-workshop on CAD/CAM. The participants were asked to fill up a questionnaire together with their feedback (in terms of value, rationale, difficulty or equations) on the spot during the seminar. The list of participants is presented in the appendix. [Table A 3.1]

The aim of this questionnaire was twofold:

a) To understand the problems faced by engineers while using the tools currently provided by CAD systems

b) To gather the functionality and productivity related enhancements desired by engineers.

This questionnaire was intended to help in identifying useful tools that could be developed as plug-in utilities for CAD systems. The seminar was related with Design for Manufacturability (DFM) concept, which focused on the various checks that can be carried out on a model to ease the manufacturing. DFM helps in identifying and rectifying the errors committed by the designers from the manufacturing point of view. After performing the checks on the model, the manufacturing becomes smooth and easier. These checks, when combined in one or more groups can serve as a useful plug-in to the solid modeling packages. The format of the questionnaire is presented in the appendix [Table A3.2]. Users' response is tabulated in the appendix. [Table A3.3]

3.4.1 Results

After analyzing the data from the table, a detailed comparative list of useful tools was prepared, which is presented in the appendix (Table A3.4). This table shows the percentage of users in each category against each tool. The four categories are: Very Useful, Occasionally Useful, Not Useful and Don't Know. Based on this, we arrive at the following conclusion:

Checks with very good response (more than 70% participants as very useful)

Through Hole	Inter Hole Distance
Through Hole	Hole – Edge Distance
Pockets, Slots, Cuts	Top and bottom edge fillets
Wall/Shell/Rib	Wall thickness/radius
Wall/Shell/Rib	Minimum wall thickness
Wall/Shell/Rib	Maximum wall thickness
Wall/Shell/Rib	Wall thickness around a hole or pocket
Wall/Shell/Rib	Wall thickness ratio
Boss	Height/boss dia ratio
Junctions	Arm thickness/Arm length
Undercuts	Draft#1: distance from parting
Undercuts	Draft#2: internal or external face

Checks with moderate response (Less than 50 % as very useful)

Through Hole	End Fillets
Through Hole	Number of Openings
Through Hole	Hole Shape
Pockets, Slots, Cuts	Pocket Shape
Pockets, Slots, Cuts	Array regularity
Pockets, Slots, Cuts	Distance from concave edge
Junctions	Number of arms
Undercuts	Number of undercuts

3.4 Summary

From the results of the industrial survey it can be seen that some checks, for example, hole diameter, wall thickness, inter hole distance, hole-edge distance etc. are very useful to the CAD/CAM users. These checks need to be performed frequently on a model. After performing the checks and taking the appropriate actions, the original model gets modified. Hence, there is a need of a plug-in, which immediately finds the topological difference between two models and displays the modification. Keeping this point in view, one of the tools named TopoDiff was identified for the prototype development. It was planned to develop this tool on top of GeoDiff, which has already been developed and mainly distinguishes the geometry of two models. In the next chapter, we will study GeoDiff in detail so that its features can be helpful in the development of TopoDiff.

4. A New Plug – in: TopoDiff

4.1 Introduction

When a model is modified, sometimes it is desirable to know what changes have taken place. In the process of modifying, some faces are changed, some are kept intact and some new faces are added. GeoDiff is a utility, which finds the differences between two B-Rep models. This tool identifies the unique features and modified features in the two models. But the limitation with this tool is, it doesn't see the topological differences. Hence, if the dimension of a hole is changed, this tool will show the faces and holes as modified, even though the topology is essentially the same. Similarly, if a part is scaled up or scaled down, GeoDiff shows the whole part as modified. In the following section, some important features of GeoDiff have been discussed followed by its limitations.

4.2 Study of GeoDiff

The GeoDiff Library provides functionality to establish differences between face sets of two boundary representation (B-Rep) solid models (body B1 and body B2). The functions of this library are intended to be used as a tool to track the changes made to the B-rep model.

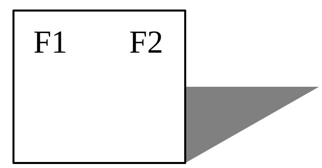
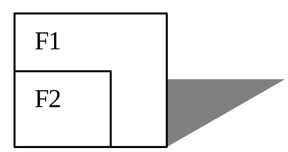
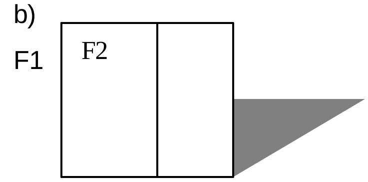
Let B1 be the solid model that is undergoing changes. Let B2 represent one such modified state of B1. The following situations may be encountered, due to the modification:

- some new faces may be added to B1
- some of the existing faces from B1 might get deleted
- the boundary of some faces will be modified
- some faces will remain unaffected by the modifications

The GeoDiff module analyzes B1 and B2 and without referring to any modification history, it identifies:

- Pairs of exactly same faces from B1 and B2
- Pairs of changed faces from B1 and B2.
- Unique faces, present only in B1
- Unique faces, present only in B2

Given a face F1 of body B1 and a face F2 of body B2, this library can process the faces and classify their relationships into the following categories:

SR NO	Description	Illustration
1	F1 and F2 are same, with complete boundary overlap	
2	F2 is completely inside F1 with partial boundary overlap.	a)  b) 

- For all interaction types, F1 and F2 share same geometry.
- Boundaries referred are external boundaries.

4.2.1 Definitions

Meanings of terms used in this documentation are described below:

Exact faces

Face F1 from body B1 and face F2 from body B2 have *exact* relationship if:

- Underlying surfaces of F1 and F2 are overlapping in 3-d space.

- Topology of F1 and F2 is exactly same.

Exact edges
Edge E1 from body B1 and edge E2 from body B2 have exact relationship if:

- Underlying curve of E1 and E2 overlap in 3-d space
- Start and end vertices of the edges match

Changed faces
Face F1 from body B1 and face F2 from body B2 have *changed* relationship if:

- Underlying surfaces of F1 and F2 are overlapping in 3-d space.
- Topology of F1 and F2 is partially same.

Changed edges
Edge E1 from body B1 and edge E2 from body B2 have *changed* relationship if:

- Underlying curve of E1 and E2 are overlapping in 3-d space
- Topology of E1 and E2 is partially same.

Unique faces
Face F1 from body B1 is said to be unique if there is no face F2, of body B2, whose extents even partially overlap that of F1.

Edge E1 from body B1 is said to be unique if there is no edge E2, of body B2, whose extents even partially overlap that of E1.

4.3 Limitations of GeoDiff

The limitations of GeoDiff are as follows:

i) Scaling: If a model is scaled up or scaled down, GeoDiff does not show the model as unchanged though the topology of the model is essentially same.
ii) Fillet Invariance: Sometimes new fillets are added or some fillets are suppressed keeping rest of the model intact. In such cases, GeoDiff Shows the model as modified, which is not desired as the filleting is solely dependent upon the manufacturing constraints.

iii) Transformation: Whenever the model is transformed (rotated, translated etc.), the topology is again essentially the same but GeoDiff shows the part as modified.

4.4 TopoDiff

TopoDiff tries to overcome the difficulties faced by GeoDiff and produces the results based on topology and not on geometry. Hence, in case of scaling, the topology is essentially same and TopoDiff will show no change in topology. If the radius of a fillet is changed, the geometry is changed but the topology is again the same. Hence, GeoDiff will show the modified faces, whereas, TopoDiff will show the topologies as before. This solves a lot of application related problems and reduces the complexity of the model. TopoDiff prepares the signature of the solid model and gives an XML file as output.

Extensible Markup Language, abbreviated XML, describes a class of data objects called XML documents and partially describes the behavior of computer programs which process them [13]. It is an application profile or restricted form of SGML, the Standard Generalized Markup Language. By construction, XML documents are conforming SGML documents. XML is designed to make it easy and straightforward to use SGML on the Web: easy to define document types, easy to author and manage SGML-defined documents, and easy to transmit and share them across the Web. XML is an abbreviated version of SGML, to make it easier for users to define their own document types, and to make it easier for programmers to write programs to handle them. It omits the more complex and less-used parts of SGML in return for the benefits of being easier to write applications for, easier to understand, and more suited to delivery and interoperability over the Web. C and C++ (and other languages like Fortran, or Pascal, or Basic, or Java or dozens more) are *programming languages* with which one specifies calculations, actions, and decisions to be carried out. On the other hand, SGML and XML are *markup specification languages* with which one can design ways of describing information, usually for storage, transmission, or processing by a program. On its own, a file of SGML or XML text (including HTML) doesn't do anything. One has to run a program to do something with it [14].

TopoDiff takes the solid model as the input and produces the signature of this model in XML format, which can be stored in appropriate folder. This file automatically gets updated when another model is used for the purpose of finding signature.

4.5 Finding Signature of the Model

The signature of the model contains information about the model, which can be used for the comparison with the signature of other model, or the same model with some modifications. After comparing the signature, one can state whether the original model has been modified or not. One important use of signature of the model is to find the properties of the model without considering the fillets, chamfers and holes as these features are normally introduced during the manufacturing process and not during design process. Hence, it would be easier for the designer to assess the model without considering the fillets, chamfers and holes. Apart from these, signature also provides important information about the mass properties, such as volume, surface area, centre of gravity etc. of the model. Following is the list of the entities provided by the signature of the model. Some of them are self-explanatory and rest are discussed below.

1. volume
2. surface_area
3. num_faces
4. num_edges
5. num_vertices
6. cen_of_gravity
7. number_of_fillets
8. number_of_chamfers
9. number_of_holes
10. no_of_fillet_invariant_faces
11. no_of_chamfer_invariant_faces
12. no_of_hole_invariant_ faces
13. no_of_chamfillhole_invariant_faces
14. no_of_concave_edges
15. avg_dist_cen_of_gravity
16. box_volume
17. diagonal_length

num_faces, *num_edges* and *num_vertices* give the number of faces, number of edges and number of vertices respectively. *cen_of_gravity* shows the centre of gravity of the model in Cartesian coordinate system.

4.5.1 no_of_fillet_invariant_faces

After adding the fillets to the model, the number of faces present in the model gets changed. For example, a cube initially has six faces but after adding two fillets to it, the total number of faces increases to eight. Each model goes for analysis after design and then it goes for manufacturing. Hence, if a model is modified, it has to go through the whole set of analysis to get qualified for manufacturing. Fillets are additional features and they are important mainly from the manufacturing point of view and hence discarding fillets at the earlier stages saves lot of efforts as well as cost. In the above case, the total number of faces without considering the fillets comes out to be six, which suggests that the model is same as the previous one (provided other properties also match) and therefore the properties of the original model can be used with this model as well.

no_of_fillet_invariant_faces calculates the total number of faces in the model without considering the fillets. *num_faces* gives the total number of faces in the model and it also creates a list for those faces. Similarly *number_of_fillets* gives the total number of fillets in the model while also generating the list of fillets. *no_of_fillet_invariant_faces* compares these two lists and remove the matching entities, resulting in a new list that contains only those faces, which are not fillets. Here is an example of the above three lists:

face_list	*fillet_list*	*fillet_invariant_faces_list*
842	854	842
844	856	844
846		846
848		848
850		850
852		852
854		
856		

Similarly, *no_of_chamfer_invariant_faces* and *no_of_hole_invariant_ faces* calculate the number of faces without considering the chamfers and the holes respectively.

4.5.2 no_of_concave_edges

Concave edges are critical from the analysis point as they are weak zones of the model. They are prone to stress concentration and also responsible for the formation of shrinkage cavity in casting process. Hence, detection of concave edges is very useful both from analysis and manufacturing point of view. Following figure shows an example of concave edge.

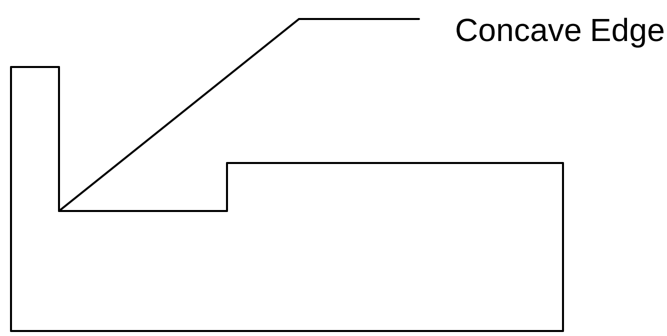


FIGURE 4.1 CONCAVE EDGE

4.5.3 avg_dist_cen_of_gravity

In some cases, the coordinates of the model changes with rotation and translation operation. To handle such cases, we need a coordinate invariant parameter to find if a model has been changed. One such parameter is *avg_dist_cen_of_gravity*, which finds the average of distances from the centre of gravity of the model to the centre of gravity of faces, without considering the fillets, chamfers and holes. This parameter is invariant as it gives the distance and not the coordinates.

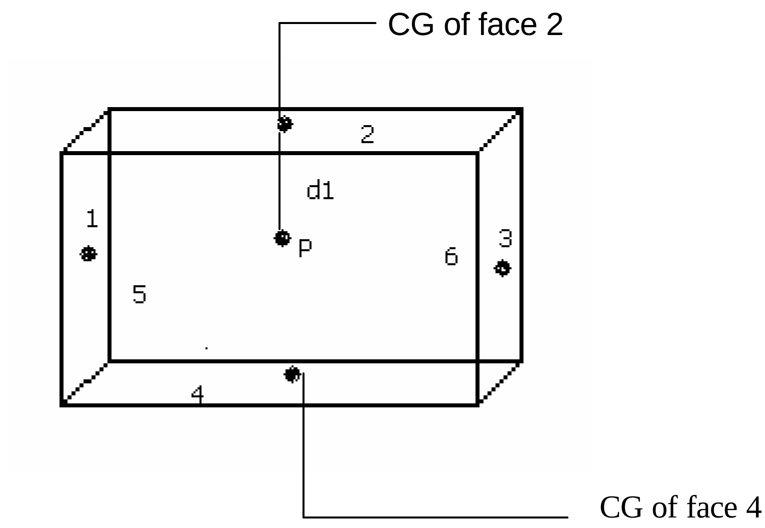


FIGURE 4.2 AN EXAMPLE OF INVARIANT PARAMETER

P is the centre of gravity of the model and 1,2,3,4,5 and 6 are the faces of the cube. d1 is the distance between the centre of gravity of the model and the centre of gravity of the face 2. Simarly, d2,d3 etc. are defined. The *avg_dist_cen_of_gravity* gives the average of these distances, which is independent of the centres of gravity of faces and the model.

4.5.4 box_volume and diagonal_length

box_volume gives the volume of the bounding box of the model. It is always more than the actual volume of the model. When only, fillets or chamfers are added to a model, the bounding box volume remains same. *diagonal_length* gives the length of the diagonal between the two extreme ends of the box. These parameters are also invariant to the coordinate of the model.

Following are the sample models and the corresponding output files:

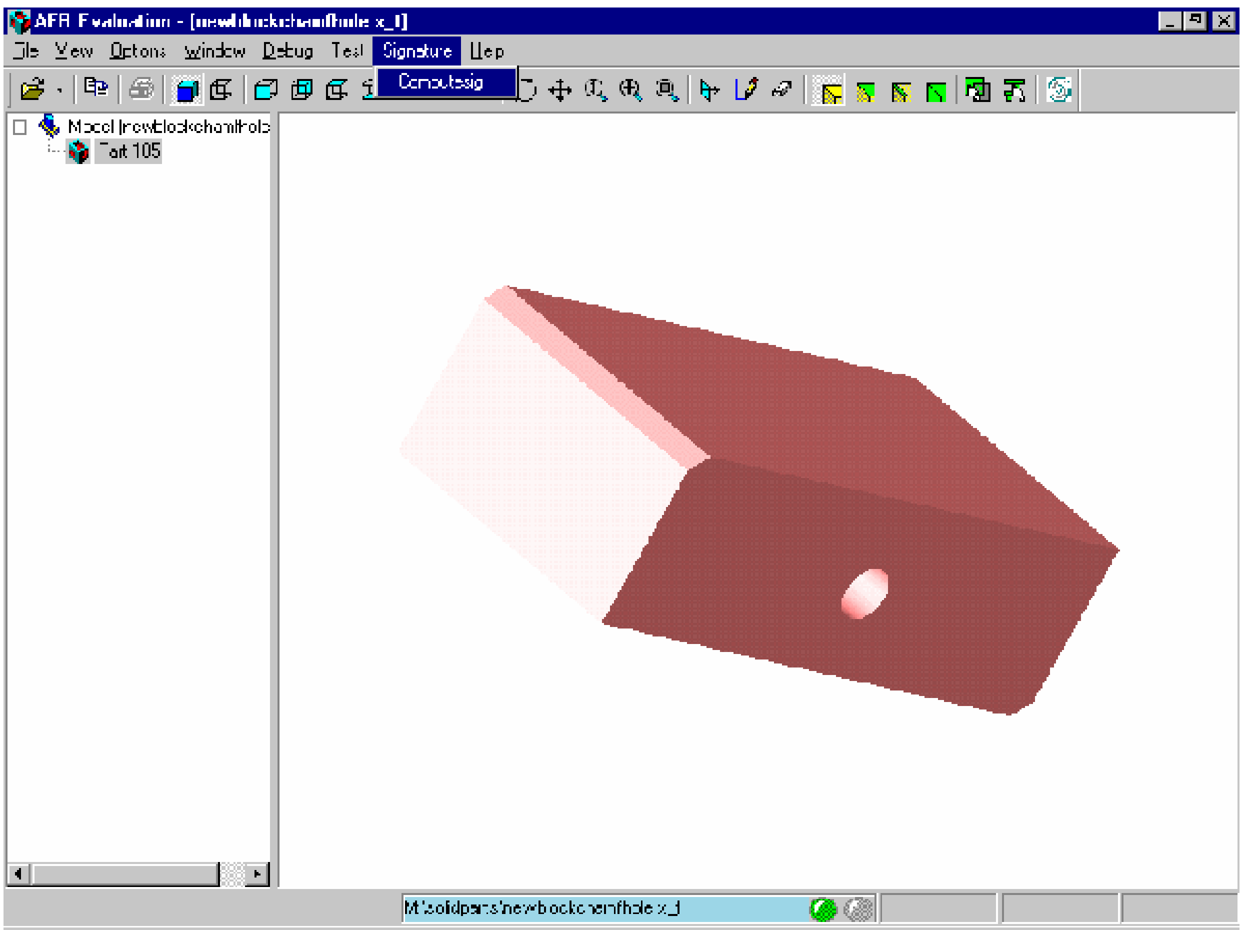
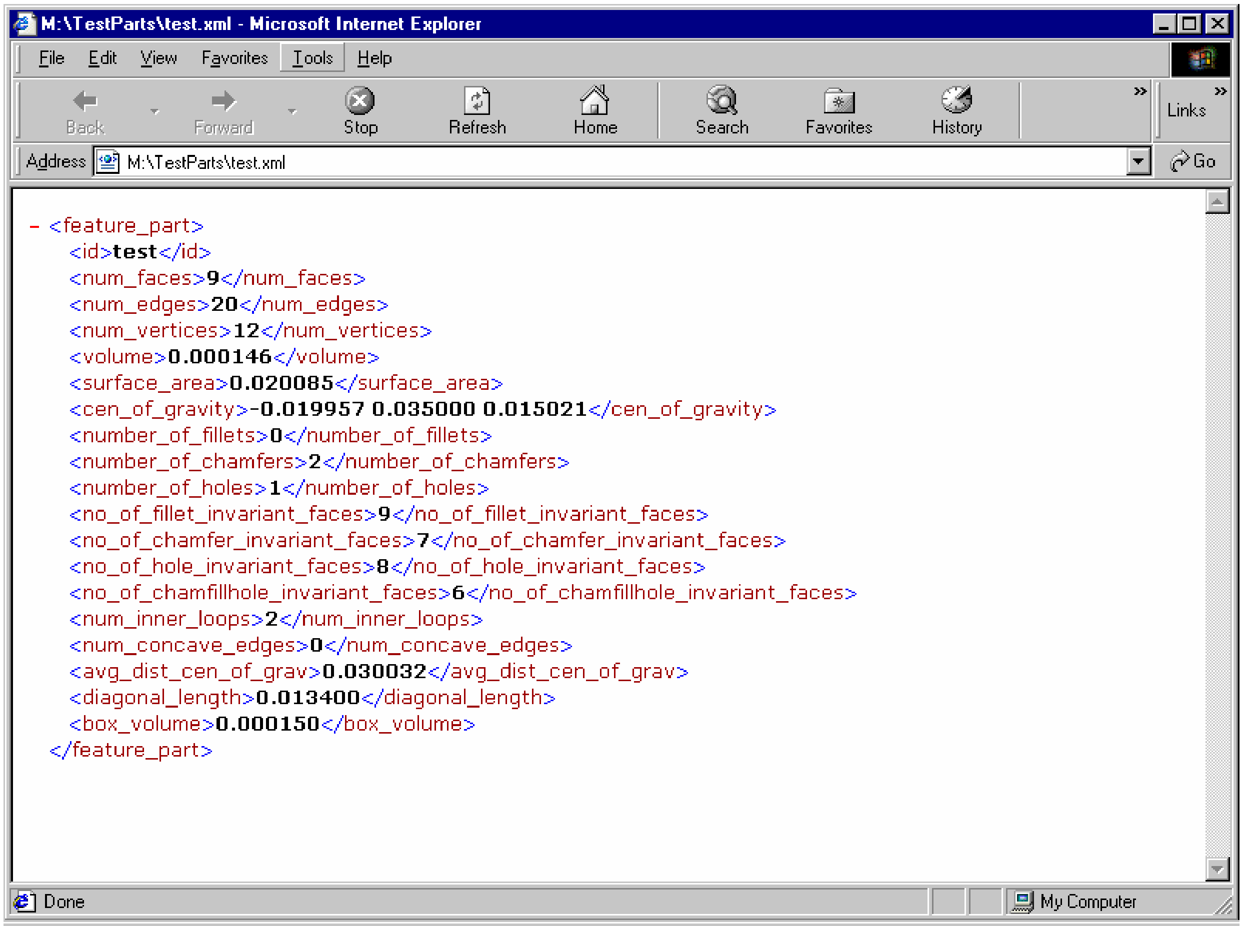


FIGURE 4.3 BLOCK WITH HOLE AND CHAMFER

XML file:




```xml
- <feature_part>
    <id>test</id>
    <num_faces>9</num_faces>
    <num_edges>20</num_edges>
    <num_vertices>12</num_vertices>
    <volume>0.000146</volume>
    <surface_area>0.020085</surface_area>
    <cen_of_gravity>-0.019957 0.035000 0.015021</cen_of_gravity>
    <number_of_fillets>0</number_of_fillets>
    <number_of_chamfers>2</number_of_chamfers>
    <number_of_holes>1</number_of_holes>
    <no_of_fillet_invariant_faces>9</no_of_fillet_invariant_faces>
    <no_of_chamfer_invariant_faces>7</no_of_chamfer_invariant_faces>
    <no_of_hole_invariant_faces>8</no_of_hole_invariant_faces>
    <no_of_chamfillhole_invariant_faces>6</no_of_chamfillhole_invariant_faces>
    <num_inner_loops>2</num_inner_loops>
    <num_concave_edges>0</num_concave_edges>
    <avg_dist_cen_of_grav>0.030032</avg_dist_cen_of_grav>
    <diagonal_length>0.013400</diagonal_length>
    <box_volume>0.000150</box_volume>
  </feature_part>
```


FIGURE 4.4 XML FILE OF BLOCK WITH HOLE AND CHAMFER

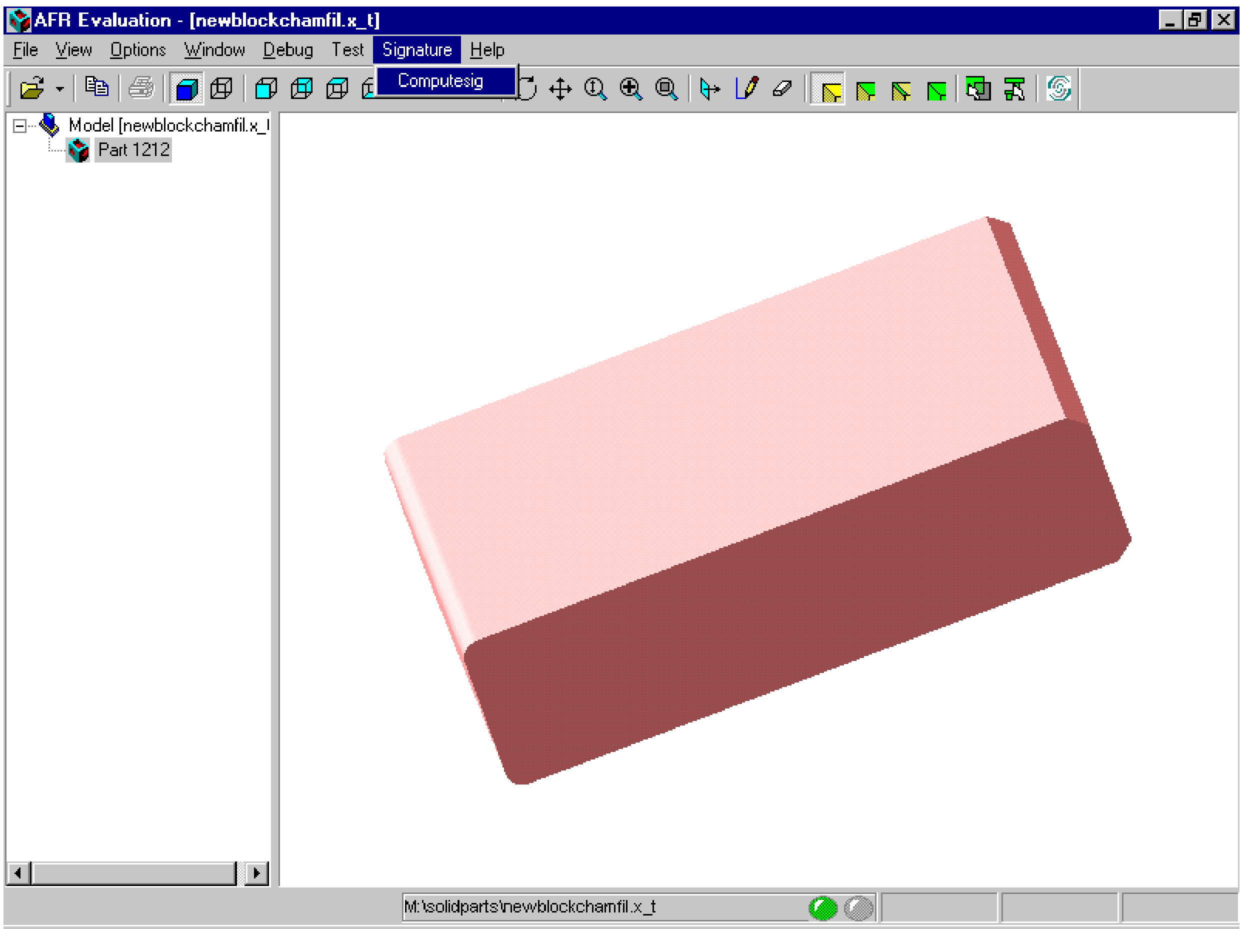


FIGURE 4.5 BLOCK WITH FILLET AND CHAMFER

XML file:

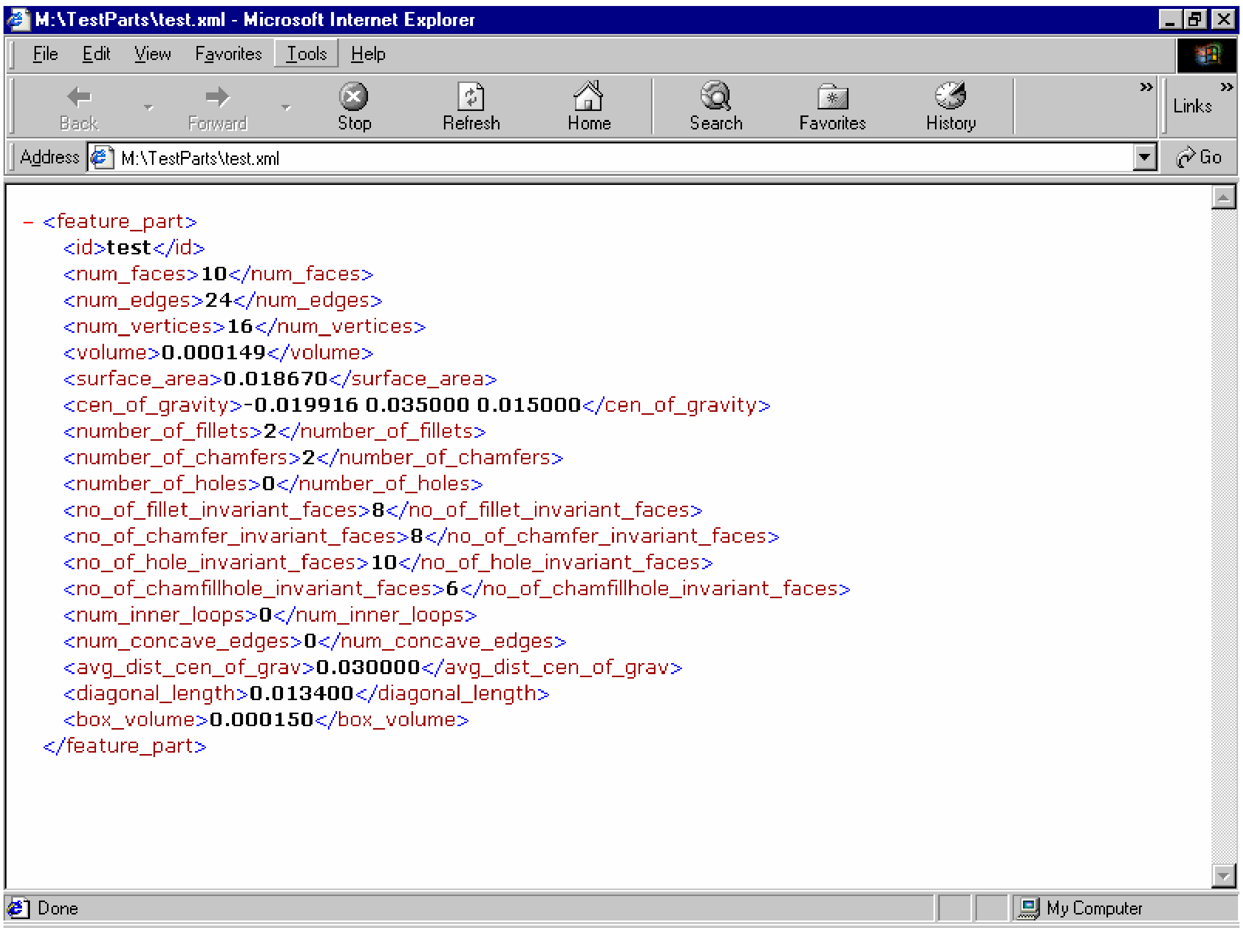


FIGURE 4.6 XML FILE OF BLOCK WITH FILLET AND CHAMFER

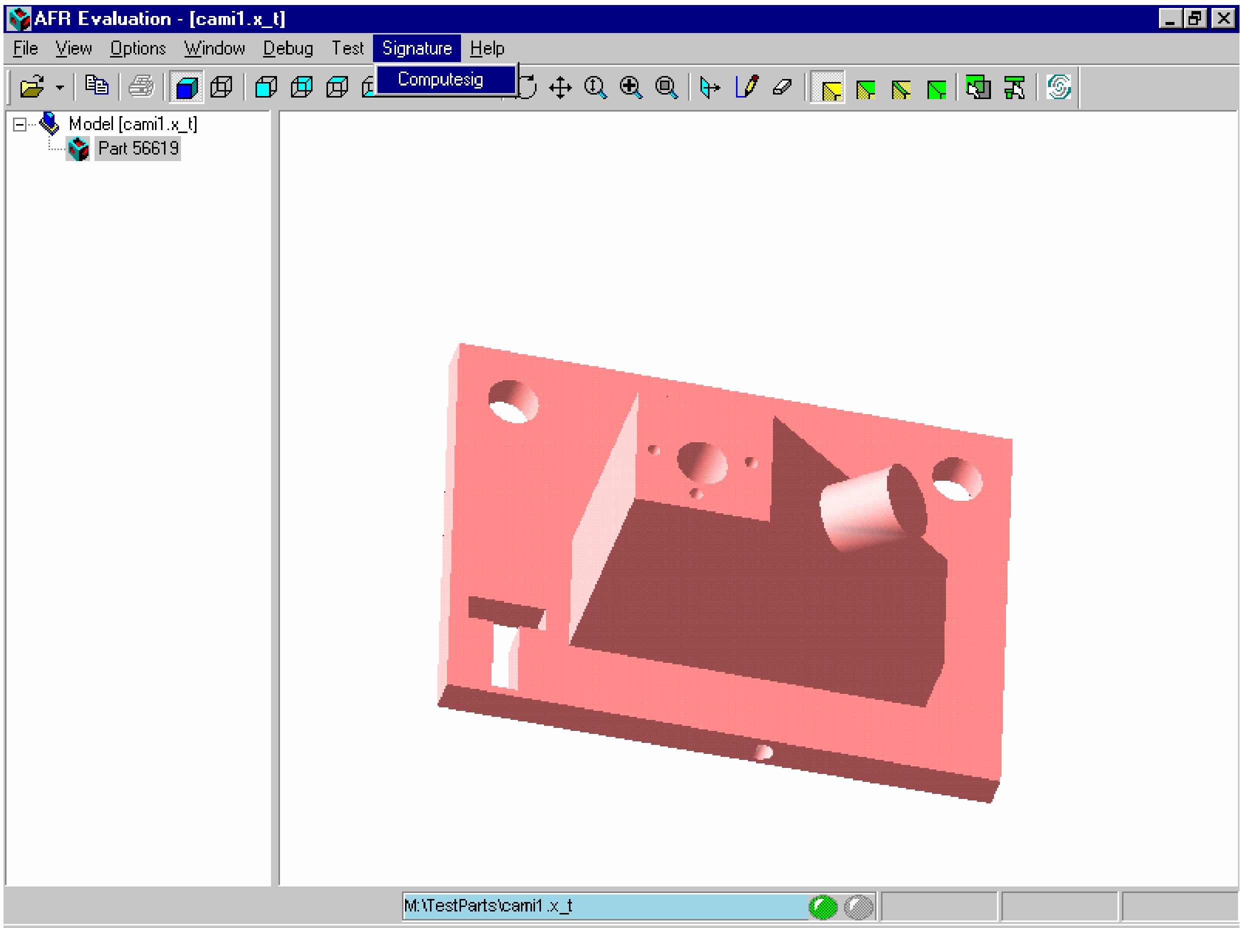
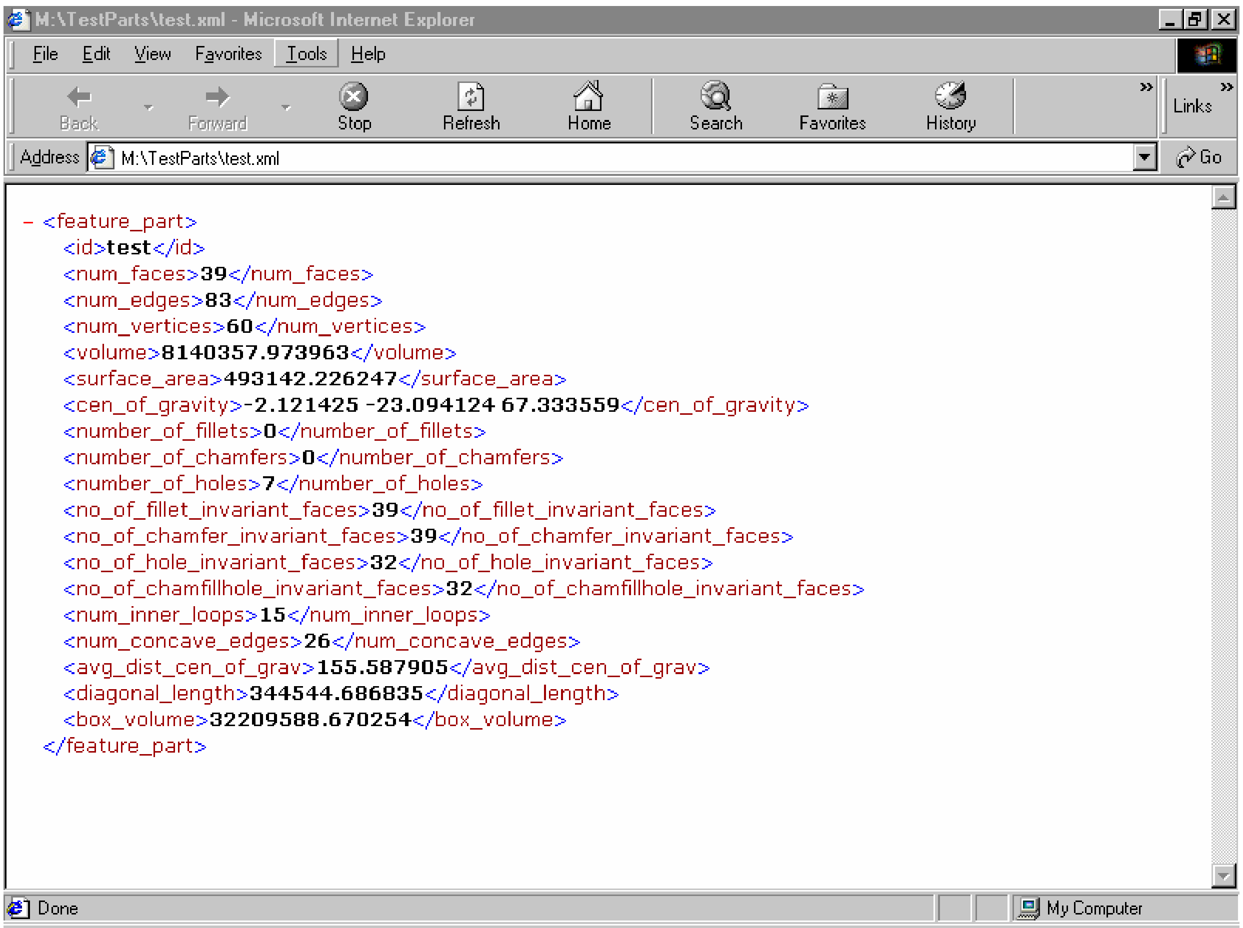


FIGURE 4.7 BLOCK WITH BOSS

XML file:




```
- <feature_part>
    <id>test</id>
    <num_faces>39</num_faces>
    <num_edges>83</num_edges>
    <num_vertices>60</num_vertices>
    <volume>8140357.973963</volume>
    <surface_area>493142.226247</surface_area>
    <cen_of_gravity>-2.121425 -23.094124 67.333559</cen_of_gravity>
    <number_of_fillets>0</number_of_fillets>
    <number_of_chamfers>0</number_of_chamfers>
    <number_of_holes>7</number_of_holes>
    <no_of_fillet_invariant_faces>39</no_of_fillet_invariant_faces>
    <no_of_chamfer_invariant_faces>39</no_of_chamfer_invariant_faces>
    <no_of_hole_invariant_faces>32</no_of_hole_invariant_faces>
    <no_of_chamfillhole_invariant_faces>32</no_of_chamfillhole_invariant_faces>
    <num_inner_loops>15</num_inner_loops>
    <num_concave_edges>26</num_concave_edges>
    <avg_dist_cen_of_grav>155.587905</avg_dist_cen_of_grav>
    <diagonal_length>344544.686835</diagonal_length>
    <box_volume>32209588.670254</box_volume>
  </feature_part>
```


FIGURE 4.8 XML FILE OF BLOCK WITH BOSS

5. Conclusion

5.1 Summary of Work Done

Various solid modeling packages such as Solid Works, Solid Edge, CATIA, Mechanical DeskTop, IDEAS etc. were studied to understand the strengths and weaknesses of these packages. Based on this study, a comparison table was prepared followed by the literature survey to explore the possibility of finding additional tools, mainly from the application-oriented industries like die & mold, automobiles, sheet metal, machine tools, etc. Industrial survey was the last step towards the identification of the plug-in to be developed. Based on the above criteria, TopoDiff was chosen to be developed as the plug-in. TopoDiff has the ability to distinguish between topology and not only geometry of two models.

5.2 Contribution

The study of solid modeling packages showed that each package has its own strengths and weaknesses. Some packages, for example IDEAS are strong in patterning, whereas some like SolidWorks and Solid Edge have better profile formation facility. Unigraphics is good at creating slots providing more alternatives like dove tail slots and ball-end slots etc. The results of the industrial survey show that some checks, for example, wall thickness-radius ratio, minimum and maximum wall thickness, inter-hole distance, arm thickness/arm length of junctions are very useful to users and if provided as a plug-in, can help in solving many application related problems. The results of TopoDiff can be used to compare one model with the other. This can be done automatically by directly feeding the data into the program. Also, we can use the signature of the model for searching similar parts in the computer.

5.3 Limitations and Future Work

The plug-in has been developed for Parasolid kernel only and hence cannot be used with other packages. However, with a slight modification in the code and project settings can result in its use in other packages as well. Since TopoDiff generates the signature of the model, it can be used for searching the similar models with specific parameters in the different directories. This would save a tremendous amount of time and also produce a sorted list of models in terms of similarities.

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[14] *http://www.ucc.ie/XML/#FAQ-ACRO*

Appendix

Table A 3.1 List of Participants

Name of the company	No. of Participants	Department
V.I.P. Industries Ltd., Nasik	3	Process Planning
		Design and Development
		Tool Engineering
Bharat Forge Ltd., Pune	4	FMD – CAD/CAM
		FMD – Engineering Department
		FMD – Engineering Department
		FMD Engineering – CAD/CAM
TELCO Automation Ltd., Chinchwad	3	SPM Design
		MTD
		Process Planning
Mahindra & Mahindra Ltd., Nasik	4	Tool Engineering
		Manufacturing Engineering
		Manufacturing Engineering
		DieShop (Press Tools Manufacturing)
Cummins India Ltd.	2	Design & Drafting
		Design & Drafting
Bajaj Auto Ltd.	4	Tool Engineering
		Tool Engineering (PDC)
		Product Engineering

		Product Engineering
Tata Ficosa Ltd.	1	Engineering
Bajaj Tempo Ltd., Pune	2	Process Engineering (CAD/CAM)
		R & D (CAD/CAM group)
Mahindra & Mahindra, Mumbai	4	Integrated Design And Manufacturing (Design)
		Integrated Design And Manufacturing Centre
		Integrated Design And Manufacturing Centre
		Integrated Design And Manufacturing (Design)
Tata Technologies (India) Ltd., Pune	6	PE – CAE
		EAG (PE)
		Process Engineering (Sheet Metal Tool Design)
		EAG (PE)
		EAG – CAD
		EAG – CAD
BEHR India Ltd.	2	Product Design
		Design
Kalyani Ltd., Pune	1	Product Development
Kinetic Engineering Limited	2	Product Design
		R & D (Product Design)

Table A 3.3 Feedback by the customers

Following is the list of checks, which were given for the feedback by the users.

No.	Check
1	Through Hole – Minimum Diameter
2	Through Hole – Aspect Ratio
3	Through Hole – Taper
4	Through Hole – End Fillets
5	Through Hole – Perpendicular Ends
6	Through Hole – Inter hole Distance
7	Through Hole – Hole-Edge Distance
8	Through Hole – Surrounding Wall thickness
9	Through Hole – Number of Openings
10	Through Hole – Hole Shape
11	Pockets, Slots, Cuts – Nom Dim/Depth
12	Pockets, Slots, Cuts: Inter – feature Distance
13	Pockets, Slots, Cuts – Pocket Shape
14	Pockets, Slots, Cuts – Thickness below pocket
15	Pockets, Slots, Cuts – Top and bottom edge fillets
16	Pockets, Slots, Cuts – Blind pockets
17	Pockets, Slots, Cuts – Array regularity
18	Pockets, Slots, Cuts: Wall-base angle
19	Pockets, Slots, Cuts – Distance from concave edge
20	Pockets, Slots, Cuts – Internal pockets
21	Wall/Shell/Rib – Wall thickness/radius
22	Wall/Shell/Rib – Minimum wall thickness
23	Wall/Shell/Rib – Maximum wall thickness

24	Wall/Shell/Rib – Wall thickness around a hole or pocket
25	Wall/Shell/Rib – wall thickness ratio
26	Boss – Height/boss dia ratio
27	Boss – Boss dia/wall thickness ratio
28	Boss – Boss height/wall thickness ratio
29	Boss – Distance to next feature
30	Boss: Inter – boss distance
31	Boss – Fillet radius/boss dia ratio
32	Junctions – Arm thickness (minimum, maximum)
33	Junctions – Arm thickness/Arm length
34	Junctions – Normal thickness/arm thickness
35	Junctions – Fillet radius at junction
36	Junctions –Number of arms
37	Junctions – Included angle
38	Undercuts – Number of undercuts
39	Undercuts – undercut volume/part volume
40	Undercuts – Dimension across and al view direction
41	Undercuts – Draft#1: distance from parting
42	Undercuts – Draft#2: internal or external face

In the following table, entities in the first row are check number (1 through 42) and entities in the first column are the participants' numbers. There are 4 values (1, 2, 3 &4) assigned to measure the usefulness of a tool.

1: If the tool is very useful

2: If the tool is occasionally useful

3: If the tool is not useful

4: If the user has no idea

No.	**1**	**2**	**3**	**4**	**5**	**6**	**7**	**8**	**9**	**10**	**11**	**12**	**13**	**14**	**15**	**16**	**17**	**18**	**19**	**20**	**21**	**22**
1	2	2	2	1	1	1	1	2	2	1	2	1	1	1	1	2	2	1	4	1	4	1
2	1	4	2	4	1	1	1	3	2	1	2	1	3	2	1	2	3	1	2	2	2	1
3	1	2	1	2	1	1	1	1	1	2	4	2	1	1	1	1	1	1	2	1	1	1
4	2	2	1	1	2	1	1	1	2	3	1	1	2	1	1	1	1	1	2	1	1	1
5	2	1	1	2	1	1	1	1	2	2	1	1	1	1	1	2	2	2	2	2	1	1
6	1	1	1	1	1	1	1	1	1	2	1	1	2	2	1	1	2	1	2	1	1	1
7	1	1	1	1	1	1	1	1	1	2	1	1	1	1	1	1	2	1	2	1	1	1
8	1	2	1	1	1	2	2	1	2	3	1	1	2	1	2	2	2	1	2	1	1	1
9	2	2	1	1	1	1	1	1	2	2	2	2	4	2	1	2	1	1	3	3	1	1
10	1	1	1	2	1	1	1	2	2	1	1	1	1	1	1	1	2	1	2	2	1	2
11	2	1	1	2	2	1	1	1	3	3	1	1	2	1	2	2	1	1	1	1	1	1
12	1	1	2	2	1	1	2	1	1	2	1	1	1	2	2	1	1	1	1	1	1	1
13	1	1	2	2	2	1	1	2	2	2	1	1	2	1	2	1	2	2	2	1	1	1
14	1	1	1	2	2	4	2	4	2	1	2	4	4	2	2	4	2	2	1	1	1	1
15	2	1	1	2	2	1	2	2	1	2	1	1	1	1	1	2	3	2	2	1	1	1
16	2	1	2	2	2	1	1	2	2	4	2	2	2	2	2	3	2	1	1	2	1	1
17	2	2	2	2	1	1	1	2	2	1	2	1	2	1	1	2	3	3	2	2	2	2
18	1	2	1	1	2	1	2	2	2	1	4	2	2	1	1	2	2	1	1	1	1	1
19	1	1	1	1	1	1	1	1	2	2	1	1	1	1	1	1	2	1	1	1	1	1
20	1	1	1	1	1	1	1	1	2	1	2	1	1	1	1	2	1	1	2	2	1	1
21	2	2	3	2	2	1	1	1	4	2	2	2	3	1	2	3	2	2	3	2	1	1
22	2	1	4	3	2	1	1	4	4	2	1	2	2	1	1	2	2	3	2	3	1	1
23	1	1	1	1	1	1	1	1	2	1	1	1	1	2	1	1	1	1	1	1	1	1
24	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
25	2	3	3	2	2	1	1	1	2	3	3	3	3	3	1	3	1	3	3	3	2	3
26	1	2	2	2	2	1	2	1	3	2	2	1	2	1	2	1	2	2	2	1	2	1
27	2	1	2	3	1	1	1	2	3	2	1	1	2	1	2	1	3	1	2	1	1	1
28	1	2	1	2	1	1	3	2	2	1	4	3	1	1	1	1	3	1	1	1	1	2

29	2	1	2	3	2	1	1	2	1	1	1	1	1	1	2	2	3	1	2	1	2	1
30	2	1	4	1	4	1	4	1	1	1	1	2	1	1	1	1	2	1	2	2	1	1
31	2	1	2	3	4	1	2	1	1	1	1	2	1	1	1	2	1	2	1	2	1	1
32	1	2	1	2	2	1	2	1	2	1	2	1	2	1	1	1	2	1	1	2	1	1
33	1	2	1	1	2	1	2	1	2	1	2	1	3	2	1	2	1	2	2	1	2	2
34	1	2	1	1	2	2	1	2	1	2	1	2	2	1	2	1	1	1	2	1	1	1
35	2	2	1	1	1	1	1	2	2	1	2	1	2	1	2	2	1	1	2	1	2	1

No.	**23**	**24**	**25**	**26**	**27**	**28**	**29**	**30**	**31**	**32**	**33**	**34**	**35**	**36**	**37**	**38**	**39**	**40**	**41**	**42**
1	1	1	1	1	4	4	1	1	2	2	1	1	1	1	1	2	4	1	1	1
2	3	1	3	2	3	3	1	2	2	2	2	3	2	2	2	2	2	2	1	1
3	2	1	1	1	1	2	2	1	1	2	1	2	2	2	1	2	1	1	1	1
4	1	1	1	1	1	2	1	1	1	1	1	2	1	1	1	2	2	2	1	1
5	1	1	2	1	1	1	1	1	1	1	1	2	1	1	1	1	1	1	1	1
6	1	1	1	2	2	2	2	1	1	1	1	1	1	1	1	2	1	2	1	1
7	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1
8	1	1	1	1	1	1	1	1	2	1	1	1	1	2	2	1	1	1	1	1
9	2	1	2	1	1	1	1	1	1	1	1	1	1	1	1	2	1	1	1	1
10	1	1	1	1	1	1	2	1	1	1	1	1	1	1	1	1	1	1	1	1
11	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	1
12	1	1	1	1	1	1	1	1	1	1	1	2	2	1	1	1	1	2	1	1
13	1	1	1	1	1	1	1	1	2	1	1	1	1	1	1	1	1	1	1	1
14	1	2	1	1	1	2	1	2	1	1	1	4	2	1	1	1	2	1	2	1
15	1	2	1	1	1	1	1	1	4	1	1	1	1	1	1	1	1	3	1	2
16	3	3	3	2	3	3	2	3	3	2	2	3	3	4	4	2	2	2	1	1
17	2	2	2	3	3	2	3	2	2	2	3	3	3	2	2	3	2	1	1	1
18	1	1	1	1	1	2	4	4	2	1	4	1	2	1	1	1	1	1	1	1
19	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	1	1	1	2
20	1	1	1	2	2	2	2	1	1	2	1	1	1	2	1	2	2	1	1	1

21	2	2	1	2	2	3	2	2	2	2	2	2	2	2	2	1	2	2	2	2
22	1	1	1	1	2	2	1	3	2	2	2	2	2	2	3	2	2	2	3	3
23	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	1	1	1	1
24	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	2	2	1	1
25	3	1	2	1	2	2	2	2	1	1	4	1	2	3	2	3	3	3	3	1
26	2	1	1	2	2	2	1	2	2	1	1	2	2	2	2	2	2	2	1	2
27	1	1	1	1	2	1	2	2	2	1	1	2	1	2	1	1	2	1	1	1
28	2	1	1	1	2	2	1	1	1	2	1	2	2	4	2	1	1	2	1	1
29	1	2	1	1	1	2	1	1	1	1	2	1	1	2	1	1	2	1	1	1
30	2	1	2	1	2	1	1	1	2	1	2	1	2	1	2	1	2	1	1	1
31	1	2	1	1	1	2	1	2	1	2	1	1	2	1	1	2	1	2	1	1
32	1	2	1	2	1	2	1	2	3	1	2	1	2	1	2	1	2	1	1	1
33	1	2	1	2	1	2	1	1	2	1	1	1	2	1	2	2	1	2	1	2
34	2	1	2	1	2	2	1	2	1	2	1	2	1	2	2	1	1	2	2	1
35	1	2	1	2	1	2	1	2	1	2	1	2	1	1	2	1	2	1	1	1

Table A 3.4 Results

Responses (in numbers and percentage) of customers

Check No.	**1** (Very useful)		**2** (Occasionally useful)		**3** Not Useful		**4** Don't know	
	No.	**%**	**No.**	**%**	**No.**	**%**	**No.**	**%**
1	22	58	16	42	0	0	0	0
2	24	63	12	32	1	2.5	1	2.5
3	23	60	11	29	3	8	1	3
4	15	39	17	45	5	13	1	3
5	22	58	16	42	0	0	0	0
6	35	92	1	3	0	0	2	5
7	29	76	7	18	1	3	1	3
8	21	55	13	34	2	5.5	2	5.5
9	14	37	17	45	4	10	3	8
10	18	47	15	39	4	11	1	3
11	22	58	10	26	1	3	3	13
12	27	71	8	21	2	6	1	2
13	18	47	13	34	5	13	2	6
14	26	68	10	26	1	3	1	3
15	27	71	11	29	0	0	0	0
16	20	53	14	37	3	8	1	2
17	12	32	17	45	8	21	1	2
18	24	63	8	21	5	13	1	3
19	13	34	18	47	4	11	3	8
20	23	61	9	24	4	10	2	5
21	31	82	6	16	0	2	1	0
22	32	84	4	11	1	2.5	1	2.5
23	27	71	7	18	3	8	1	3
24	32	84	4	10	1	3	1	3
25	32	84	4	10	2	6	0	0
26	30	79	7	18	1	3	0	0
27	23	61	10	26	3	8	2	5
28	20	53	13	34	3	8	2	5
29	26	68	10	26	1	3	1	3
30	26	68	9	24	2	5	1	3

31	24	63	12	32	2	5	0	0
32	24	63	11	29	0	0	3	8
Check No.	**1** (Very useful)		**2** (Occasionally useful)		**3** Not Useful		**4** Don't know	
	No.	**%**	**No.**	**%**	**No.**	**%**	**No.**	**%**
33	27	71	8	21	0	3	2	5
34	20	53	10	26	0	8	5	13
35	22	58	12	32	3	8	1	2
36	19	50	13	34	1	3	5	13
37	24	63	11	29	1	3	2	5
38	19	50	13	34	5	13	1	3
39	20	53	15	39	2	6	1	2
40	22	58	12	32	3	8	1	2
41	31	82	4	10	2	5	1	3
42	32	84	5	13	2	3	0	0

1 - 10 Through Hole

11 – 20 Pockets, Slots And Cuts

21 – 25 Wall/Shell1

26 – 31 Boss

32 - 37 Junctions

38 – 42 Undercuts

Checks with very good response (more than 70% participants as very useful)

Through Hole – Inter Hole Distance

Through Hole – Hole – Edge Distance

Pockets, Slots, Cuts – Top and bottom edge fillets

Wall/Shell/Rib – Wall thickness/radius

Wall/Shell/Rib – Minimum wall thickness

Wall/Shell/Rib – Maximum wall thickness

Wall/Shell/Rib – Wall thickness around a hole or pocket

Wall/Shell/Rib – wall thickness ratio

Boss – Height/boss dia ratio

Junctions – Arm thickness/Arm length

Undercuts – Draft#1: distance from parting

Undercuts – Draft#2: internal or external face

Checks with moderate response (Less than 50 % as very useful)

Through Hole – End Fillets

Through Hole – Number of Openings

Through Hole – Hole Shape

Pockets, Slots, Cuts – Pocket Shape

Pockets, Slots, Cuts – Array regularity

Pockets, Slots, Cuts – Distance from concave edge

Junctions –Number of arms

Undercuts – Number of undercuts

Checks with poor response (More than 10 % as not useful)

Through Hole – End Fillets

Through Hole – Number of Openings

Through Hole – Hole Shape

Pockets, Slots, Cuts – Pocket Shape

Pockets, Slots, Cuts – Array regularity

Pockets, Slots, Cuts: Wall-base angle

Pockets, Slots, Cuts – Distance from concave edge

Undercuts – Number of undercuts

Checks with less knowledge among users (more than 10% as don't know)

Pockets, Slots, Cuts – Nom Dim/Depth

Junctions – Normal thickness/arm thickness

Junctions –Number of arms