Enhance Selection Intent to Simplify Selection of Edges for Rib,Draft and Blending workflow in NX-CAD Software Development

By

Shirsale Govinda Anil

10MMCC17

DEPARTMENT OF MECHANICAL ENGINEERING

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Major Project

Submitted in partial fulfillment of the requirements

For the degree of

Master of Technology in Mechanical Engineering

(CAD/CAM)

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Declaration

This is to certify that

i) The thesis comprises my original work towards the degree of Master of Technology in Mechanical Engineering (CAD/CAM Engg.) at Nirma University and has not been submitted elsewhere for a degree.

ii) Due acknowledgement has been made in the text to all other material used.

Govinda Anil Shirsale
10MMCC17

Undertaking for Originality of the Work

I, **Shirsale Govinda Anil**, Roll. No.**10MMCC17** , give undertaking that the Major Project entitled "**Enhance Selection Intent to Simplify Selection of Edges for Rib,Draft and Blending workflow in NX-CAD Software Development**" submitted by me, towards the partial fulfillment of the requirements for the degree of **Master of Technology in Mechanical Engineering(CAD/CAM Engg.)** of Nirma University, Ahmedabad, is the original work carried out by me and I give assurance that no attempt of plagiarism has been made. I understand that in the event of any similarity found subsequently with any published work or any dissertation work elsewhere; it will result in severe disciplinary action.

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This is to certify that the Major Project entitled **"Enhance Selection Intent to Simplify Selection of Edges for Rib,Draft and Blending workflow in NX-CAD Software Development"** submitted by **Shirsale Govinda Anil (10MMCC17)**, towards the partial fulfillment of the requirements for the degree of **Master of Technology in Mechanical Engineering (CAD/CAM Engg.)** of Nirma University of Science and Technology, Ahmedabad is the record of work carried out by him under my supervision and guidance. In my opinion, the submitted work has reached a level required for being accepted for examination. The results embodied in this major project, to the best of my knowledge, haven't been submitted to any other university or institution for award of any degree or diploma.

Prof. J M Dave
Guide, Professor,
Department of Mechanical Engineering,
Institute of Technology,
Nirma University, Ahmedabad

Mr. Amit Inamdar
Guide, Software Engineering Associate Manager,
NX-Part Modeling Team,
Siemens (PLM) Software Industry Ltd.,
Hinjewadi IT Park,Pune

Dr. R N Patel
Head and Professor,
Department of Mechanical Engineering,
Institute of Technology,
Nirma University, Ahmedabad

Dr K Kotecha
Director,
Institute of Technology,
Nirma University, Ahmedabad

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- Shirsale Govinda Anil
10MMCC17

Abstract

This project is intended to make it easier to Draft and Blend rib structures in a part model. It will implement the ability to easily collect the external edges of a group of faces. 'Curve' Selection Intent will be enhanced to add two new rules: "First Rule" and "Second Rule".

"First rule" allows the collection of edges by selecting faces, either directly or by selecting the features that created, modified, or collected those faces. Edges that are common to faces selected under this rule will be removed, i.e., all the faces selected are treated as a single face, and the external edges of that single face are selected.

The another new rule "Second rule" is essentially the "First rule" with additional edge removal for simplifying the drafting or blending of ribs (or rib-like structures). In this the edges are categorized as Concave and Convex, under this rule major set of convex edges are ultimately collected for the faces selected.

Under these new rules, faces can be selected by selecting face-creating features from either the graphics window or the Part Navigator.The 'Curve Collector' block will be modified to add the new 'First rule' and 'Second rule' to the other rules that it already offers.

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Chapter 1

Introduction

1.1 Problem Statement

The 'Face Edges' Selection Intent rule supports features such as 'Draft' and 'Edge Blend' for the selection of edges. Although this rule allows multiple faces to be selected, they can only be selected one-at-a-time. Besides requiring numerous face picks, this method also collects the edges that are shared by adjacent faces. The commands (e.g., Draft) that do not know how to process these shared edges, fail when multiple faces are picked.(see fig 1.1)

Users need to be able to easily select multiple faces using a 'Curve' rule, but they do not want the rule to select the shared edges. For example, currently if a user wants to draft from all the top edges of the part shown in the following figure, there is no efficient way to select them.

Using 'Face Edges'rule currently requires five face picks (each face must be selected individually). An additional problem with selecting multiple faces, is that the rule collects the edges between the faces (shared edges). For the 'Draft (From Edges)' and 'Edge Blend' commands, the user does not want the shared edges to be collected when multiple faces are selected under the 'Face Edges'rule.(see fig 1.2)

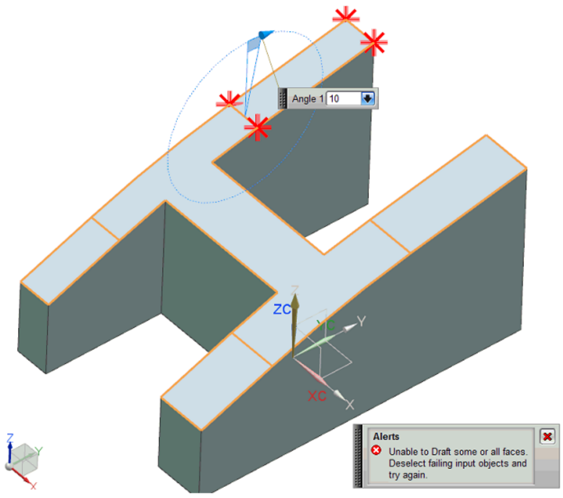

Figure 1.1: Defective Part

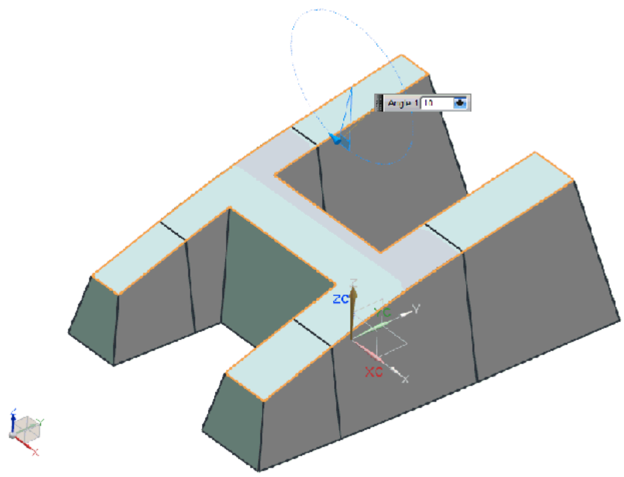

Figure 1.2: Defect Free Part

1.2 Project in Brief

This project is intended to make it easier to Draft and Blend rib structures. It will implement the ability to easily collect the single-convexity, perimeter edges of a group of faces. The faces may be selected directly and/or through the selection of features that created faces. Essentially the many faces selected are considered to be one face,

and it's like-convexity, bounding edges are collected.
The another new rule "Second rule" is essentially the "First Rule" rule with additional edge removal for simplifying the drafting or blending of ribs (or rib-like structures). In this the edges are categorized as Concave and Convex, under this rule major set of convex edges are ultimately collected for the faces selected.
The convexity filtering is an option, and can be suspended. This provides flexibility for possible use in other commands.

1.3 Project Scope

The scope of the current project is limited to the implementation of new selection intent rules 'First rule' and 'Second rule' for 'Curve Collector' block and new rules will only be enabled in following two features in NX:

- Draft feature
- Edge Blend feature

The 'First rule' will allow the user to collect the perimeter edges of a group of faces, by selecting the faces, or by selecting features that created the faces. This rule must also allow the selection of the 'Group Face' feature .The user will be able to select features both graphically, and from the Part Navigator.

The 'Second rule' will behave similar to 'First rule' additionally it allow selecting only 'like convexity' edges from Perimeter edges.

Under the new rules, Under Deselecting, the 'All of Intent' option will remove the particular edge being deselected, and also remove all edges that were added with it, including the seed.[2]

Chapter 2

Literature Survey

2.1 Evolution of CAD/CAM

In today's fast paced world, CAD/CAM systems have become an essential element in manufacturing companies through out the world. Technology and communication are changing rapidly, driving business methods for organizations and requiring capitalization in order to maintain competitiveness. Knowledge prior to investing into a system is crucial in order to maximize the benefits received from changing CAD/CAM systems. The CAD /CAM market place has seen a significant new advance in the technology every 7 years and is ripe for the emergence of a major new development.

During the past 20 years, the computer-aided design (CAD) industry has simplified software user interfaces to enable engineers to design products more quickly and efficiently. The purpose of the session was to discuss how CAD software might be made more productive in the future. During the past 20 years, the computer-aided design(CAD) industry has simplified software user interfaces to enable engineers to design products more quickly and efficiently[5].

CAD is sometimes translated as "computer-aided design", "computer-aided drafting", or a similar phrase. A related acronym, CADD, stands for "computer-aided design and drafting". All these latter terms are essentially synonymous, and refer to the designing and technical drawing of various projects on a computer rather than a traditional drawing board. The spectrum of engineering projects commonly created with computer-aided drafting is broad, and include architectural drafting, mechanical drafting, electrical drafting, and other forms of design communication. Today they constitute part of a broader definition of computer-aided design. In this session we will mention at MCAD (mechanical compute raided design).In general Computer Aided Design (CAD) package has three component : a)Design , b) Analysis, c) Visualization. A brief description of these component :

- Design :- Refers to geometric modeling, 2-D and 3-D modelling ,including, drafting, part creation, creation of drawings with various views of the part, assemblies of the parts, etc.
- Analysis :- Analysis refers to finite element analysis, optimization and other number crunching engineering analysis. In general a geometric model is first created and then the model is analyzed for loads, stresses, moment of inertia and volume etc.
- Visualization :- Visualization refers to computer graphics , which include : rendering a model, creation of pie charts, contour plots, shading a model, sizing, animation etc. The visualization is a presentation of the final model to the interested. Each of these three areas has been extensively developed in the last 30 years. Several books are written on each of these subjects and courses are available through the academic institutions and the industry.

2.2 The capabilities of modern CAD systems

- Reuse of design components
- Ease of design modification and versioning
- Automatic generation of standard components of the design

- Simulation of designs without building a physical prototype
- Automated design of assemblies, which are collections of parts and/or otherassemblies
- Output of engineering documentation, such as manufacturing drawings, andBill of Materials
- Validation/verification of designs against specifications and design rules
- Output of design directly to manufacturing facilitiesDevelopment in CAD resulted in the following tools and methods
- Wireframes, solid modelling
- Intelligent wiring diagrams and production linked database systems
- Graphically represented system or plant diagrams and databases
- Parametric design models
- Real-time process simulation

2.3 Solid Modeling

Programs that are capable of solid modeling can be much more powerful than simple wireframe modelers. These programs are used to build parts that are actually solid objects instead of simply a wireframe outline of the part. Since these parts are represented as solids, they have volume, and if given a density can have a weight and mass as well. The computer can calculate many physical properties of these parts, such as center of gravity and moments of inertia. These calculations can even be performed for irregularly shaped parts, for which manual calculations would be extremely difficult. Finite Element Analysis techniques can also be used to perform stress analysis of these parts.[1]

The weakness of wireframe and surface modeling is that they have Ambiguous and incomplete geometric description, lack topological information, tedious modeling process and awkward user interface. Solid modeling is based on complete, valid and unambiguous geometric representation of physical object:

- Complete :- Points in space can be classified. (Inside/ outside)
- Valid :- vertices, edges, faces are connected properly.
- Unambiguous :- there can only be one interpretation of object

2.3.1 Solid Modeling Methods

There are two basic methods used to create solid models. They are Constructive Solid Geometry (CSG) methods, and Boundary Representation (Brep) methods. CSG uses solid primitives (rectangular prisms, spheres, cylinders, cones, etc.) and Boolean operations (unions, subtractions, intersections) to create the solid model. B-rep methods start with one or more wireframe profiles, and create a solid model by extruding, sweeping, revolving or skinning these profiles. The Boolean operations can also be used on the profiles themselves and the solids generated from these profiles. Solids can also be created by combining surfaces, which often have complex shapes, through a sewing operation.

These two methods can often be combined in order to create the desired parts. Each of these methods has its limitations, and parts which are very difficult to create using just one or the other method can be created much more easily using a combination of both methods. Thus, most commercial solid modeling systems are hybrids using both CSG and Brep methods.

2.3.2 Constructive solid geometry (CSG)

In CSG objects are represented as a combination of simpler solid objects (primitives). The primitives are cube, cylinder, cone, torus, sphere etc. Copies or instances of these primitive shapes are created and positioned. A complete solid model is constructed by combining these instances using set specific, logic operations (Boolean).

Each primitive solid is assumed to be a set of points, a boolean operation is performed on point sets and the result is a solid model. Boolean operations include union, intersection and difference. The relative location and orientation of the two primitives have to be defined before the boolean operation can be performed. Boolean operation can be applied to two solids other than the primitives. Boolean operations includes -Union, Difference, Intersection.

a. Union :- The sum of all points in each of two defined sets. (Logical OR).Also referred to as Add, Combine, Join, Merge.

b. Difference :- The points in a source set minus the points common to a second set. (Logical NOT) Set must share common volume Also referred to as subtraction, remove, cut.

c. Intersection :- Those points common to each of two defined sets (logical AND).Set must share common volume. Also referred to as common, conjoin Boolean operation is intuitive to user, easy to use and understand and provide for the rapid manipulation of large amounts of data. Data structure does not de

   ne model shape explicitly but rather implies the geometric shape through a procedural description. Object is not defined as a set of edges and faces but by the instruction: union primitive1 with primitive 2 .This procedural data is stored in a data structure referred to as a CSG tree. The data structure is simple and stores compact data and is also easy to manage.

2.3.3 CSG tree

CSG tree stores the history of applying Boolean operations on the primitives. Stores in a binary tree format. The outer leaf nodes of tree represent the primitives. The interior nodes represent the Boolean operations performed. More than one procedure (and hence database) can be used to arrive at the same geometry.

2.3.4 Advantages of CSG

a. CSG representation is unevaluated .Faces, edges, vertices not de

ned in explicit

b. CSG model are always valid Since built from solid elements.

c. CSG models are complete and unambiguous

d. Easy to construct a solid model minimum step.

e. CSG modeling techniques lead to a concise database so less storage.

f. Complete history of model is retained and can be altered at any point.

g. Can be converted to the corresponding boundary representation.

2.3.5 Limitations of CSG

a. Only Boolean operations are allowed in the modeling process...With Boolean operation alone, the range of shapes to be modeled is severely restricted thus not possible to construct unusual shape.

b. Requires a great deal of computation to derive the information on the boundary, faces and edges which is important for the interactive display/ manipulation of solid.

2.4 Feature Based Modeling and Parametric Modeling

2.4.1 Feature Based Modeling

Dimension-driven design refers to a collection of solid-modeling capabilities that include parametric, and feature based. Feature-based modeling has, among engineers, rapidly become the preferred method of constructing solid models. In feature-based packages, solid models are constructed from geometric features such as slots, shells, bends, drafts, rounds, and so forth. The alternative is to construct models using mathematical geometric entities such as unions of spheres, cylinders, and boxes. One advantage of features is that they provide dimensions that correctly de

ne how the feature behaves when dimensions change.

For hole drilled through a plate, in a feature-based modeling system, the geometry has enough embedded intelligence to know that the hole should go all the way through the plate, regardless of how thick the plate is. Thus, even if the designer decides to increase the plate thickness tenfold, the hole will still go through to the other side.

In a model defined with older solid geometry schemes, the designer would have to manually lengthen the hole if the plate became thicker. Otherwise, the hole would stop within the plate. The formal way of referring to this property is that a feature is capable of producing many different geometric instances, depending on the dimension values that the designer spells out. The most important aspect of feature-based techniques is that they capture design intent. In the drilled hole example, the designer intended to put a hole through the plate. This intent was maintained regardless of what changes were made in the plate dimension. Another important property of feature-based modelers is the ability to let a feature reference the geometry of various models in an assembly. This referencing allows changes made in one model to propagate to other affected models. One example is where a metal housing has features that are dimensioned from other parts mounted to the frame. When these parts move or change shape, the housing updates as well.

Feature-based models have been likened to a recipe approach to building solid models. Once a design has been specified, it is possible can be created in simple tables. This offers the opportunity to create generic designs. For example, adding a macro pro- gram that prompts for inputs would enable the creation of a customized solid-model assembly, complete with drawings and related data such as a bill of materials, cutter paths, and so forth. Feature-based modelers allow operations such as creating holes, fillets, chamfers, bosses, and pockets to be associated with specific edges and faces. When the edges or faces move because of a regeneration, the feature operation moves along with it, keeping the original relationships. The choices made developing these models are very important. If the features aren't referenced correctly, they may not end up in the correct place if the model is regenerated. A feature that is located at an X and Y offset from a corner of the face instead of at the center of the face will not remain at the center of the face when the model is regenerated unless constraints are added to the model that will change the X and Y offsets to keep the feature at the center of the face.

2.4.2 Parametric Modeling

Parametric methods depend on the sequence of operations used to construct the design. The software maintains a history of changes in specific parameters. The point of capturing this history is to keep track of operations that depend on each other, so that, whenever it is told to change a specific dimension, the system can update all operations referenced to that dimension.

For example, a circle representing a bolt hole may be constructed so it is always concentric to a circular slot. If the slot moves, so does the bolt circle. [1] Parameters are usually displayed in terms of dimensions or labels and serve as the mechanism by which geometry changes. The designer can change parameters manually by changing a dimension, or by referencing them to a variable in an equation that is solved by the modeling program itself. Parametric modeling is most efficient in working with designs where changes are likely to consist of dimensional changes rather than grossly different geometries.

Important feature of modern CAD systems is the ability to create parametric models. In a parametric model, each entity, such as a Boolean primitive, a line or arc in a wireframe, or a filleting operation, has parameters associated with it. These parameters control the various geometric properties of the entity, such as the length, width and height of a rectangular prism, or the radius of a fillet. They also control the locations of these entities within the model.

These parameters can be changed by the operator as necessary to create the desired part. Parametric modelers that use a history-based method keep a record of how the model was built. When the operator changes parameters in the model and regenerates the part, the program repeats the operations from the history, using the new parameters, to create the new solid. There are many uses for this type of modeling. Designers can test various sizes of parts to determine which is the best part for their use by simply adjusting the model parameters and regenerating the part. Some parametric modelers also allow constraint equations to be added to the models. These can be used to construct relationships between parameters.[6]

Chapter 3

NX CAD Modeling System

3.1 Introduction to NX CAD

NX is the commercial CAD/CAM/CAE software suite developed by SIEMENS. It is commonly referred to as a 3D PLM software application. All stages of product development are supported, from conceptualization, to design (CAD), to analysis (CAE), to manufacturing (CAM).

NX was previously known as Siemens. NX is a parametric solid / surface feature based package based on Parasolid. NX is widely used throughout the engineering industry, especially in the automotive and aerospace sectors. NX can provide users with industry specific tools known as wizards to help streamline and automate complex processes, as well as improve product quality and user productivity. For example, the Mold Wizard module addresses the specific requirements of plastic injection applications while the Progressive Die Wizard addresses the specific requirements of die stamping applications.[2]

3.2 NX CAD Modeling

NX Modeling provides a solid modeling system to enable rapid conceptual design. The solid models which are complex, realistic can be created and edited interactively. The solid bodies can be changed and updated by directly editing their dimensions or by using other construction techniques.

3.2.1 Sketch

Use the Sketcher to freehand a sketch, and dimension an "outline" of curves. The solid or sheet body can be created by sweeping the sketch using Extrude or Revolved Body. This sketch can be refined to precisely represent the object of interest by editing the dimensions and by creating relationships between geometric objects. Editing a dimension of the sketch not only modifies the geometry of the sketch, but also the body created from the sketch.

3.2.2 Creating and Editing Features

Use features in creating models such as holes, slots and grooves. The dimensions of feature can be directly edited as per requirement and locate the feature with dimensions. For example, a Cylinder is defined by its diameter and height. All the parameters of cylinder can be edited by entering new values.

3.2.3 Associativity

Associativity is a term that is used to indicate geometric relationships between individual portions of a model. These relationships are established as the designer uses various functions for model creation. In an associative model, constraints and relationships are captured automatically as the model is developed. For example, in an associative model, a through hole is associated with the faces that the hole penetrates. If the model is later changed so that one or both of those faces moves, the hole updates automatically due to its association with the faces.

3.2.4 Positioning a Feature

The positioning dimensions are used to position a feature relative to the geometry on model. The feature is then associated with that geometry and will maintain those associations whenever the model is edited. The position of a feature can be edited by changing the values of the positioning dimensions.

3.2.5 Reference Features

The reference features, such as Datum Planes, Datum Axes and Datum CSYS can also be created, which can be used as reference geometry when needed, or as construction devices for other features. Any feature created using a reference feature is associated to that reference feature and retains that association during edits to the model. The datum plane can be used as a reference plane in constructing sketches, creating features, and positioning features. Similarly datum axis can be used as reference axis to create datum planes, to place items concentrically, or to create radial patterns.

3.2.6 Expressions

The Expressions tool lets us incorporate the requirements and design restrictions by defining mathematical relationships between different parts of the design. For example, the height of a boss can be defined as three times its diameter, so that when the diameter changes, the height changes also.

3.2.7 Boolean Operations

Modeling provides the following Boolean operators: Unite, Subtract, and Intersect. Unite combines bodies, for example, uniting two rectangular blocks to form a T-shaped solid body. Subtract removes one body from another, for example, removing a cylinder from a block to form a hole. Intersect creates a solid body from material shared by two solid bodies. These operations can also be used with free form features called sheets.

3.2.8 Undo

The Undo function is used to return to the previous state of design any number of times. It is not necessary to take a great deal of time making sure each operation is absolutely correct, because a mistake can be easily undone. This freedom to easily change the model lets us cease worrying about getting it wrong, and frees us to explore more possibilities to get it right.

3.2.9 Additional Capabilities

Other NX applications can operate directly on solid objects created within Modeling without any translation of the solid body. For example, like in drafting, engineering analysis and NC machining functions by accessing the appropriate application. Using Modeling, a complete, unambiguous, three dimensional models can be designed to describe an object. The wide range of physical properties can be extracted from the solid bodies, including mass properties.

Shading and hidden line capabilities help to visualize complex assemblies. The interferences can be automatically identified, eliminating the need to attempt to do so manually. Hidden edge views can later be generated and placed on drawings. Fully associative dimensioned drawings can be created from solid models using the appropriate options of the Drafting application. If the solid model is edited later, the drawing and dimensions are updated automatically.

3.2.10 Parent/Child Relationships

If a feature depends on another object for its existence, it is a child or dependent of that object. The object, in turn, is a parent of its child feature. For example, if a HOLLOW (1) is created in a BLOCK (0), the block is the parent and the hollow is its child.

A parent can have more than one child, and a child can have more than one parent. A feature that is a child can also be a parent of other features. To see all of the parent-child relationships between the features in our work part, open the Part Navigator.

3.2.11 Creating a Solid Model - A Simple Example

Modeling provides the design engineer with intuitive and comfortable modeling techniques such as sketching, feature based modeling, and dimension driven editing. An excellent way to begin a design concept is with a sketch. When sketch is used, a rough idea of the part becomes represented and constrained, based on the

t and function requirements of design. In this way, design intent is captured. This ensures that when the design is passed down to the next level of engineering, the basic re- quirements are not lost when the design is edited. The strategy which is used to create and edit model to form the desired object depends on the form and complexity of the object.

The next several figures illustrate one example of the design process, starting with a sketch and ending with a finished model. First, sketch outline of curves can be created. Then this sketch can be swept or rotated to create a complex portion of the design.

Finally, form features can be added, such as chamfers, holes, slots, or even user de

ned features to complete the object.

3.3 NX-CAD Feature Modeling

Feature-based modeling has, among engineers, rapidly become the preferred method of constructing solid models. In feature-based packages, solid models are constructed from geometric features such as holes, fillets, chamfers, bosses, and pockets to be associated with specific edges and faces. The alternative is to construct models using mathematical geometric entities such as unions of spheres, cylinders, and boxes. One advantage of features is that they provide dimensions that correctly de

ne how the feature behaves when dimensions change.

With the use of computer vision how feature can be to suit for parametric based modeling. Parametric methods depend on the sequence of operations used to construct the design. The software maintains a history of changes in specific parameters. The point of capturing this history is to keep track of operations that depend on each other, so that, whenever it is told to change a specific dimension, the system can update all operations referenced to that dimension. For example, a circle representing a bolt may be constructed so it is always concentric to a circular slot. If the slot moves, so does the bolt circle.

The term "Feature" is used generally in NX to describe a class of objects that have defining parents. A feature's parents enable it to recall the inputs and the operation that were used in its creation. Features include all solids, bodies, primitives and certain wireframe objects.[3]

Features can be described by the following characteristics:

a. The inputs of a feature are its "parents" and the resulting feature object is the "child," which is Associative or "associated with" its parents.

b. Parents can be geometric objects or numerical variables (known as Expressions). In the case of numerical variables, the numbers are known as "parameters" of the child object, and the child is said to be "parametric."

c. If any object is modified, its associated children are updated (regenerated) to reflect the change.

d. The combination of parents and the creation operation is sometimes known as the "history" of an object.

e. The parent-child analogy can be extended further within NX, and it is reasonable to speak of ancestors, descendents, siblings, orphans, reparenting and so on. (See Parent/Child Relationships.)

3.3.1 Other Common Terms Used in Feature Modeling

Table 3.1: Common Terms Used in Feature Modeling

Body	**A class of objects containing solids and sheets.**
Solid body	A collection of faces and edges that enclose a volume.
Sheet Body	A collection of one or more faces that do not enclose a volume.
Face	A region on the outside of a body enclosed by edges.

3.3.2 Common Concepts

Table 3.2: Common Concepts in NX software

Object Selection	**Throughout all Feature options we are required to select objects.**
Defining Points	All points, including origin points, limit points, start points, and endpoints are defined using the Point Con- structor.
Target Solid	The solid body on which we create new features. If there is only one solid body displayed, the system selects the target solid for us. Otherwise, we must select the body we want to identify as the target.
Boolean Operations	When we create primitives and swept features, we must choose to either create a new target solid or perform a Boolean operation with respect to an existing target solid.
Defining Vectors	All directions, reference, and destination vectors are defined using the Vector Constructor. Features created using the Thru option of Face Association define vectors in I, J, K components mapped to the Absolute Coordinate System. We can enter I, J, K components as real values or expressions.

Chapter 4

Design Overview

The new 'First rule' will be designed to be Multi Entity rule. For each selection made by user one 'First rule'is created. All first rules then grouped under one 'Multi Entity' rule. Edge length filtering will be part of the 'Second rule' and will be done as post processing on output of 'First rule'contained in it.

4.1 Object Design

we know that there are many rules in any CAD software to select the specific entities in a graphic user interface. The importance of these rules is, they help the user to pick all or specific entities in a single pick.The entities that are selectable under specific rules is called seed of that particular rule.

Now consider the several rules from NX CAD software.

- Single Curve :- This rule is used to collect all single curves/Edges of a solid model from graphic window.
- Connected Curves :- This rule is used to collect all connected curves/Edges of a solid model from graphic window.
- Tangent Curves :-This rule is used to collect all Tangent curves/Edges of a solid model from graphic window.

- Face Edges:-This rule is used to collect all Edges of a face/faces of a solid model from graphic window.
- Feature Curves :- This rule is used to collect all curves/Edges of feature from graphic window.

So similarly here my objective is to implement selection intent rules for some specific features in NX CAD Software.

4.2 General Procedure To Implement Rules

Here i am using Face Edges rule to explain the General procedure.

- As C++ is the coding language used for any CAD software introduce a new class for the new rules. Class is nothing but a collection of member functions and member variables.
- First important task is to prepare input data or Rule data for the new rule.As discussed earlier the input data involves the seed means the type of entities to be collected with that rule.So for Face Edges to collect all edges user can select Feature,Face and Solid Body.
- Enable the rule in User interface, so that it is visible to the user.
- Customize the rule for different feature,means here we can activate the rule for all or some specific feature like Blend,Draft,Extrude,Trimmed Sheet.
- Next step is to implement core method to perform our desired task.For example in Face Edges rule suppose user selects Feature, then we write one method to process the input "feature" and collect all the edges of that feature and then highlight all the Edges of that feature.
- Besides this main task new rule should support other functionalities like Copy paste,Deselection,RMB.[2]

Chapter 5

Algorithms and User Interface

5.1 Algorithm

Activity Diagram for First Rule.

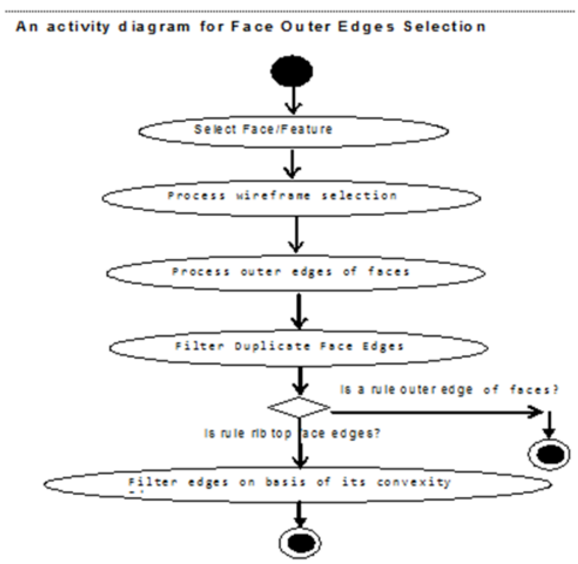

Figure 5.1: Activity Diagram

5.2 Computational Algorithms

- Select Feature/Face
- Process selection list

    Process selection

- check entity type and call corresponding rule's process method, for our two rules call
- ProcessEdges which

    Collects selected entities

    Collects Edges rejected by client

    Return tracks to the caller

    Add entities rejected by client in to selection list

    Add negated entities in to selection list

    Return tracks to the caller

- Create collector from selection list

    Create rules from selection list

    Create collector

    Replace rules in collector

        Create a unique list of all the negated entities

        Create selection intent rule

        Replace rule

- Initialize input data for outer face edges rule

- Update and evaluate edges

    Filter Duplicate edges

    If rule is "Second rule" do filtering

      Return tracks to the caller

- Return tracks to the caller
- Exit

5.3 GUI Interactions

Two new rules will be added in rule selection dropdown list," Second rule " and " First rule". These two rules will also be available for RMB for selected Face/Feature. User interface will look like this.

5.3.1 Rules in selection Intent Drop down List

First rule Second rule Rules after doing RMB on selected Face/Feature

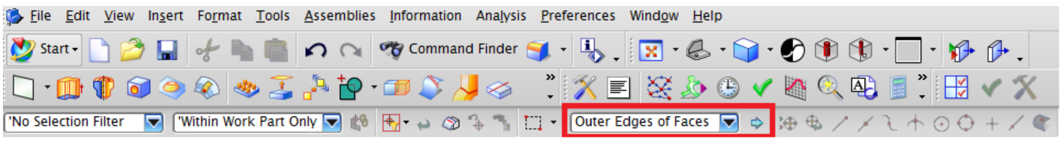

Figure 5.2: First Rule

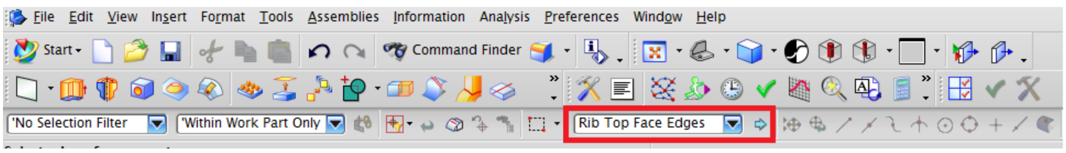

Figure 5.3: Second Rule

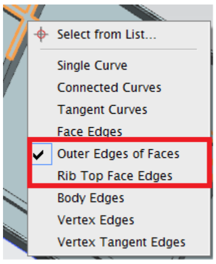

Figure 5.4: Rules after doing RMB on selected Face

Chapter 6

Results And Conclusion

6.1 Functional Testcases

Functional Testcases is a key to check and validate the result.A test case consist of data which instructs the tester that what feature he has to test, what are the inputs and what are the expected outputs. If the actual output of the feature matches with the expected output we say the test case is passing. Following are the test cases used for testing.

Test Type	**User Interface**
Title	Verify new First Rule and Second Rule in Curves Selection Intent for Draft Feature
Test Case Instruction	1. Create new part file 2. Invoke Draft feature by Insert-Detail Feature-Draft 3. Set type as from edges 4. Check New Rule First rule and Second rule" in selection Intent block for Draft Edge Selection
Validation	1. The Curve selection intent drop down should have new rule's, in addition to the existing rules for Draft feature.

Test Type	**User Interface**
Title	Verify new rule's in Curves Selection Intent for Edge Blend Feature
Test Case Instruction	1. Create new part file 2. Invoke Edge blend feature by Insert-Detail Feature-Edge Blend 3. Check New Rules in selection Intent block for Edge to blend.
Validation	1. The Curve selection intent drop down should have new rule's, in addition to the existing rules for Edge Blend feature 2. Check that existing Rules are also available for Edge Blend features

Test Type	**Functional**
Title	Features will be selectable from the Part Navigator for draft feature using "Second rule"
Test Case Instruction	1. Open given part 2. Try to Create new draft feature using Second rule. 3. In draft dialog , set Type as from Edges. 4. For draw direction, set vector as positive Z axis direction. 5. For stationary edges selection, Set Second rule. 6.Select Group face from Part Navigator, by selecting feature Group face from P.N. 7. Enter the draft angle 5 and click on ok button of dialog
Validation	1. Selection of Features for new rules should be selected from Part navigator. 2. Edges that are common to faces selected under this rule will be removed. 4.Feature should be created successfully using new rule.

Test Type	**Functional**
Title	Features will be selectable from the Part Navigator for draft feature using ”First rule”
Test Case Instruction	1. Open given part 2. Try to Create new draft feature using First rule 3. In draft dialog , set Type as from Edges 4. For draw direction, set vector as positive Z axis direction. 5. For stationary edges selection, Set First rule. 6. Select Group face feature from Part Navigator 7. Enter the draft angle and click on ok button of dialog.
Validation	1. Selection of Feature for new rules should be selected from Part navigator. 2. Edges that are common to faces selected under this rule will be removed. 3.DraftFeature should be created successfully using new rule

Test Type	**Functional**
Title	Group Face Features Will be Selectable :User will actually select faces by selecting Group Face Features that collect faces ('Group face').
Test Case Instruction	1. Open given part. 2. Try to Create new draft feature using Second rule. 3. In draft dialog , set Type as from Edges. 4. For draw direction, set vector as positive Z axis direction. 5. For stationary edges selection, Set Second rule. 6. Select faces of Group face from graphics selection,by select group face from quick pick list. 7. Enter the draft angle and click on ok button of dialog.
Validation	1. Selection of faces for new rules, Group faces and /or direct selection of faces should be selected for edge blend and draft features. 2. Edges that are common to faces selected under this rule will be removed.

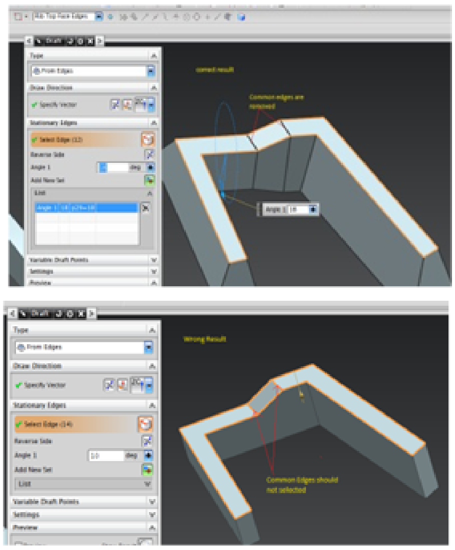

Figure 6.1: Reference figure

Test Type	**Functional**
Title	Automatic Removal of Shared Edges between Rules (selection of multiple features) for Draft feature using new Second rule.
Test Case Instruction	1. Open given part. 2. Invoke draft command. 3. Set type as from edges. 4. Set Draw direction as positive Z axis. 5. For Stationary Edges selection, Set Second rule and select multiple feature by selecting Divide. face,Group face,Offset face,Group face features from window graphics selection.
Validation	1. Feature selection should works for new rules. 2. Edges that are common to Features selected under this rule will be removed.

Test Type	**Functional**
Title	Automatic Removal of Perimeter of Faces that have smallest combined length, when some perimeter edges are convex and some perimeter edges are concave used on rib-like structure for draft feature.
Test Case Instruction	1. Open given part. 2. Invoke draft command. 3. Set type as from edges. 4. Set Draw direction as positive Z axis. 5. For Stationary Edges selection, Set Second rule and select top faces, (Green Face as show in image) from window graphics selection. (This face has convex and concave edges). 6. Enter the draft angle. 7. Click on Ok button. 8. Check the result for Edge selection for Combined length of Convex edge is smallest than Concave edge.
Validation	1. For condition Combined length of Convex edge is smallest than Concave edge then Convex edge should be remove. 2. Shared Edges should be remove. 3.draft Feature should be created successfully using new rule.

6.2 Conclusion

The two new selection intent rules 'First rule' and 'Second rule' implemented successfully for Draft feature and Edge Blend feature.so now the 'First rule'allow the user to collect the perimeter edges of a group of faces, by selecting the faces, or by selecting features that created the faces. This rule also allows the selection of the 'Group Face' feature .The user is able to select features both graphically, and from the Part Navigator.

The 'Second rule' behave similar to 'First rule' additionally it allow selecting only 'like convexit' edges from Perimeter edges. Under the new rules, Under Deselecting, the 'All of Intent' option is enabled to remove the particular edge being deselected, and also remove all edges that were added with it, including the seed.

Thus we can conclude the due to above mentioned changes and enhancements the design time of the designer has been reduced and the accuracy of work has increased, and the user experiences consistency in various features of NX. Also due to above enhancements robustness of the NX software is maintained irrespective of its version.

6.3 Future Scope

Now the two new selection intent rules 'First rule' and 'Second rule' implemented for Collector based features (this includes Edge Blend feature and Draft feature),in future based on special requirements these two rules can also be enabled for Section based features(this includes Extrude Feature,Divide Face Feature,Trimmed Sheet Feature,Instance Geometry Feature,and Project Curve Feature).

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