PARAMETRIC MODELING FOR MECHANICAL COMPONENTS

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Abstract: parametric modeling is a technique to generalize specific solid model. This generalization of the solid model is used to automate modeling process. Here for example a case study of valve component is discussed. The single control variable can reduce complexity and ambiguity in parametric updates. Feature based modeling approach also used to show reduction in complexity and ambiguity of parametric modeling.

Key words: part family, template part, instantiate part, topology, design history, feature modeling, feature suppression, feature reorder.

1. INTRODUCTION

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. If several parameters always require the same value, or a certain parameter depends on the values of several others, this is the best way to ensure that these relationships are always correct.

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. PROBLEM DEFINITION:

As presented in previous studies, there are several ways in which variability of a parameterized model can be implemented, including

• The use of design tables stored as ASCII files or as Excel file

• The use of built-in scripting capabilities of a CAD system

• The use of API for invoking CAD functions within programs

The use of design tables is attractive, especially if implemented using Excel files. The wealth of mathematical functions available in Excel can be readily employed, for example to generate random sets of values. Such capabilities can be also used through specially developed programs.

With this technology, designers can concentrate their efforts on the overall design and gradual refinement of the details. If a design solution is novel or a specific case, usually the range of solutions offered by commercial packages does not cover such novel solution. The designer must invert invest effort customizing the application and building new libraries that on time will narrow new design solutions for the same designer. Flexibility for maintain control of the design is a must for parametric modeling.

The progression of operations using hierarchies and allowing changing parameters is a useful tool considering different design options and real time visualization. In addition, this register can serve as basis for analysis of the design process and the generation and use of knowledge by designers, becoming a tool for improve performance in design process and a knowledge repository. A more efficient ways to simplify the process of constraint definition imply reviewing the redundancy of operations. For instance, requirement resulting from design and fabrication processes can be contradictory.
However the main limitation of this method is that it does not in itself provide the means to test the validity of generated models, which must be implemented by other means. The use of built-in scripting is an attractive solution. Scripts can be written to perform various functions, including checking of the geometry for validity.

3. METHODOLOGY:

In the design phase, modelers express object behavior in terms of explicit parameters and geometric constraints. The resulting parametric objects will often be hierarchical assemblies, with a main object controlling the geometry of constituent objects. The assembly may be self-contained (i.e. all of the relationships are internal, with the overall geometry driving the geometry of constituent objects), or they may be dependent on (one or more) other objects (such as a window inserted in a wall adjusting to the thickness of the wall). Parametric relations basically act as constraints on a model and control the degree of freedom of models. Parametric models may be under, fully-, or over-defined. For example, when the minimum number of dimensions required for fully defining a shape is provided, the state of the shape is called fully-defined.In the implementation phase, the translated object behavior is implemented in a CAD system as a parametric model. If a CAD system provides a good design interface, the design phase and the implementation phase can be done iteratively.
In the validation phase, the implemented parameters and geometric constraints should be checked against the descriptions of initial design intent and should be optimized. The semantic validity of a model can only be judged by domain expertise. Incorrect (or 'absurd') design situations are obvious to a human viewer, but are amorphous and thus very difficult to identify algorithmically.

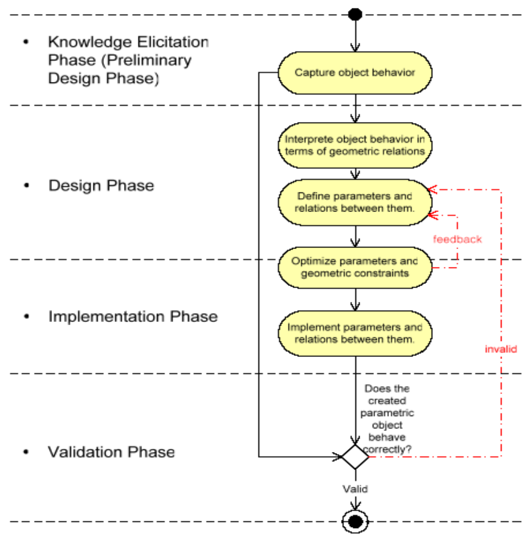

Figure 1.basic parametric modeling process

The current version of CAD-Gate operates on Unigraphics part files, which are, in general assemblies. Each component of an assembly has its own parameter list. If it is required to change more files of the assembly, one has to run CAD-Gate as many times as necessary to change each of the required files.

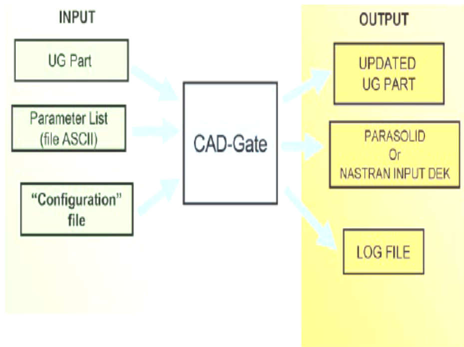

Figure 2. UG/NX parametric modeling configuration

Figure 3. constraints list

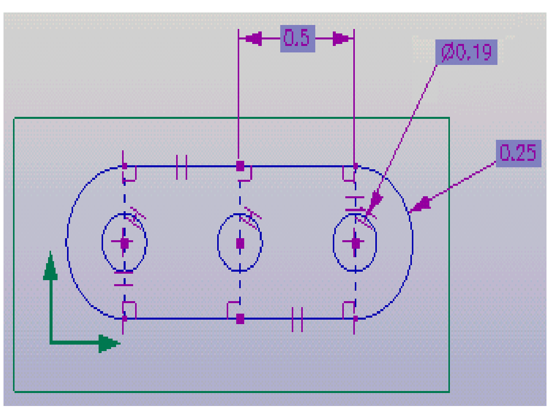

Figure 4. constrained sketch

Live Rules allows automatic recognition of relationships and maintains them throughout an edit, even if there were no defined rules. Many geometric conditions are "obvious" to a user, but since traditional CAD systems don't understand this, relationships must be defined to maintain those conditions

In the image below we show a dimensioned part. Note that the dimensions are in the 3D model and that they are easy to interpret. Changing any of these dimensions changes the part!

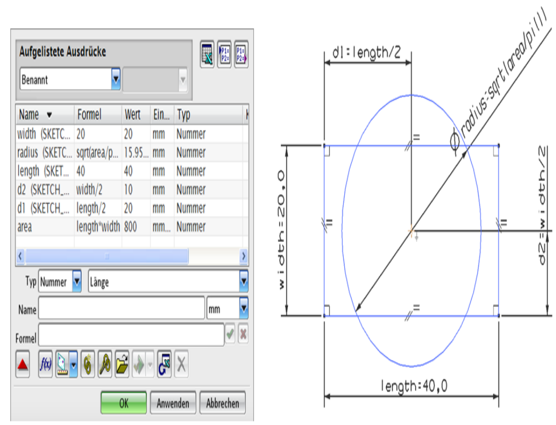

Figure 5. parameter table for sketch

Look at the part modification above, it was changed by selecting the radius of the outer diameter and changed. The concentric relationship of the hole was automatically recognized because that relationship was found by Live Rules. While the edit may appear to be simple. The complexity is in the connected tapered faces which must remain tangent to the boss cylinder as the hole is changed.

In UGS NX, the parametric part modeling process involves the following steps:

1. Set up Units and Part name.
2. Determine the type of the base feature, the first solid feature, of the design. Note that Extrude, Revolve, or Sweep operations are the most common types of base features.
3. Create a rough two-dimensional sketch of the basic shape of the base feature of the design.
4. Apply/modify constraints and dimensions to the two-dimensional sketch.
5. Transform the parametric two-dimensional sketch into a 3D solid.
6. Add additional parametric features by identifying feature relations and complete the design.
7. Perform analyses/simulations, such as finite element analysis (FEA) or cutter path

generation (CNC), on the computer model and refine the design as needed.

8. Document the design by creating the desired 2D/3D drawings.

Here are some general guidelines for creating sketches in UGS NX:

• Create a sketch that is proportional to the desired shape. Concentrate on the shapes and forms of the design.

• Keep the sketches simple. Leave out small geometry features such as fillets, rounds and chamfers. They can easily be placed using the Fillet and Chamfer commands after the parametric sketches have been established.

• Exaggerate the geometric features of the desired shape. For example, if the desired angle is 85 degrees, start by creating an angle that is 50 or 60 degrees (and make adjustment later). Otherwise, UGS NX might assume the intended angle to be a 90-degree angle.

• Draw the geometry so that it does not overlap. To create a 3D features from a 2D sketch, the 2D geometry used should define a clear boundary. Self-intersecting geometry shapes are not allowed, as it cannot be converted into a solid feature.

• The sketched geometric entities should form a closed region. To create a solid feature, such as an extruded solid, a closed region is required so that the extruded solid forms a 3D volume.

4. A CASE STUDY OF VALVE MODEL

Various valve model are studied here as a case study for parametric modeling. Various types of valves are modeled such as needle valve, butterfly valve, ball valve etc.

Following figure shows various models of valve in UG/NX.. the mechanical valve are considered here due to its variation in shape and feature.

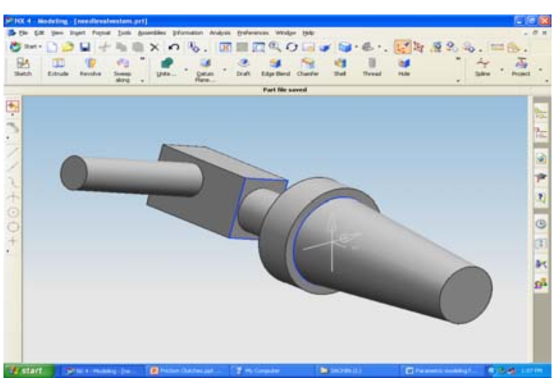

Figure 6. valve model in UG/NX

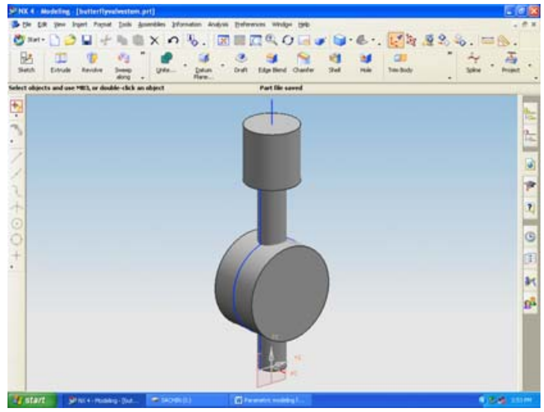

Figure 7. butterfly valve stem model in UG/NX

Expression toolbar for valve model: In expression toolbar of UG/NX software all parameter defining geometry are defined and formulated to single control variable. Thus parametric updates become less complex.

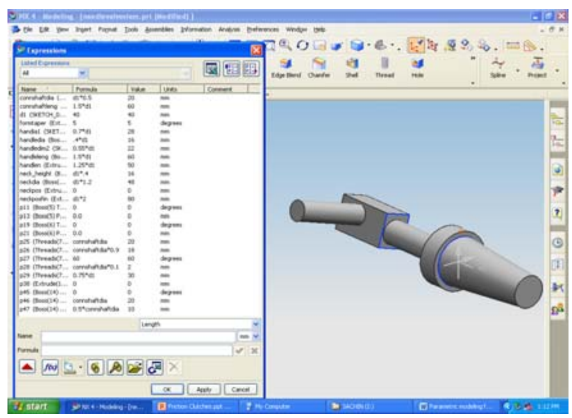

Figure 8. expression toolbar for needle valve stem

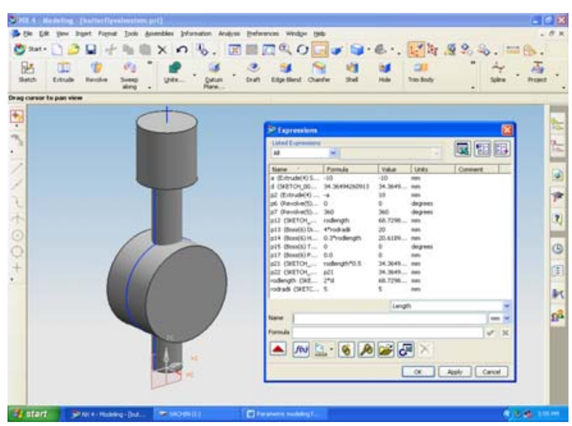

Figure 9. expression toolbar for butterfly valve stem

Part family toolbar for valve model:Part family toolbar in UG/NX software defines attributes of master part or generic part. The modeling software has inbuilt excel interface. This excel sheet interface facilitates easy updates of parameters. The schematic diagram for part family concept is shown in fig. 10

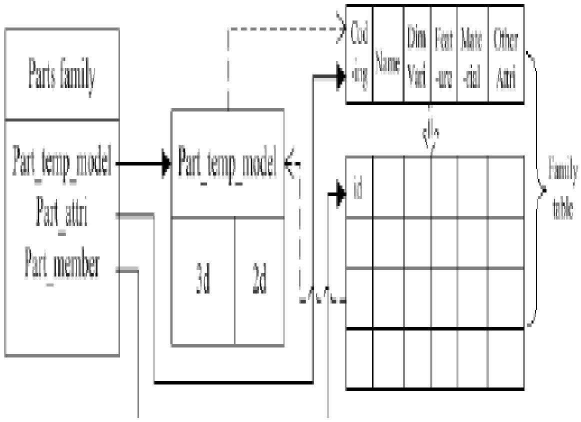

Figure 10. schematic representation of part family in UG/NX

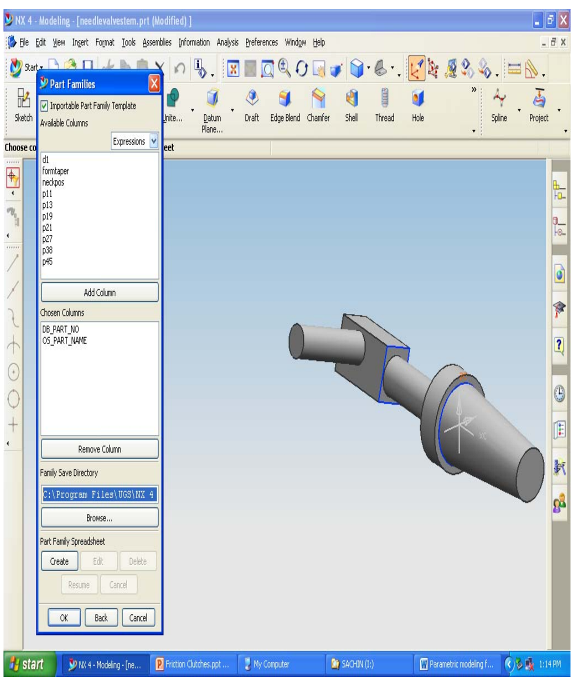

Figure 11. part family tool for needle valve stem in UG/NX

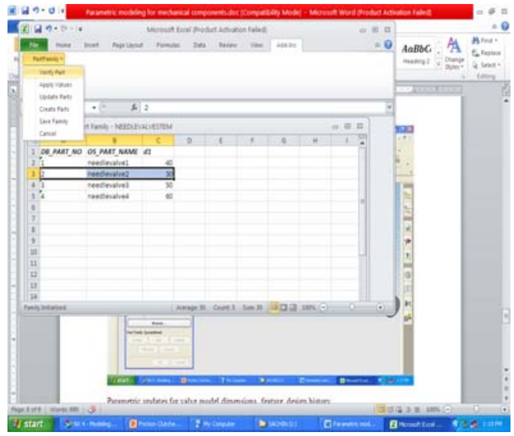

Figure 12. excel interface of UG/NX

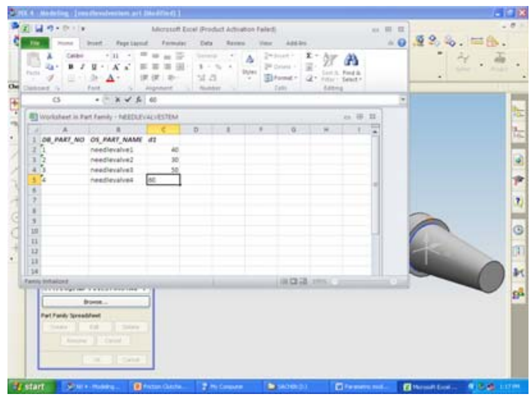

Figure 13. excel interface of UG/NX

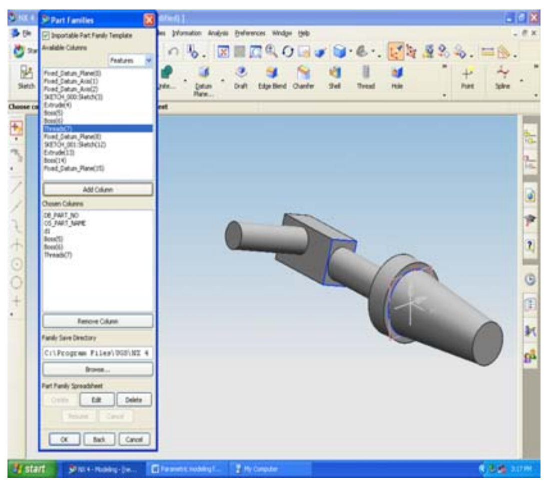

Figure 14. excel interface of UG/NX

5. PARAMETRIC CAD CHALLENGES

Simplest of changes requires typically a deep understanding of the design procedure.

Imported CAD data is often treated as "dumb" information. Traditional parametric CAD solutions often lack the ability to make changes to non-native CAD data Interoperability.

Inability to implement late-stage design changes; design intent limits the ability to implement unexpected design changes. Slow and tedious process to understand and rework design intent, so that models must be re-created.

Analysts, who are not expert CAD users, struggle to modify parametric CAD geometry, thus they must rely on CAD experts to de-feature and prepare models for analysis.Model history and design intent limit the ability to make changes to designs.

The parametric 3D design process is often too slow for rapid prototyping. Difficulty in working with and editing legacy data or data from multiple CAD tools

6. CONCLUSION:

Parametric modeling for valve components is considered here. It is obvious that this method is more user friendly method of modeling automation. The various approaches of parametric modeling automation are discussed like dimensions, constraints to basic datum plane, feature based modeling,

Capturing ideas within parametric technology can result in a faster design process, the result being a completed design in significantly less time than a conventional CAD system. parametric technology makes design relationships results in building initial designs faster, and even more important, since most designs are an iterative process, not needing complex modeling makes editing designs easier, resulting in an shorter overall design process.

7. BENEFITS

Model creation: Virtually command free creation means less steps in creating a model, yielding a faster design. Adding intent during or after creation using 3D model constraints, 3D driving dimensions, or adding variable equations proves to be easier and more flexible. By not ordering features, but instead collecting them, gives a fast iteration capability, even orders of magnitudes faster than regenerating history-based modeling. By eliminating the engineering time involved in determining how to edit history-based complex models, the result should be less engineering time and not avoiding editing existing models or even rebuilding models. Users should examine their own experiences to judge potential savings in model editing for ECO's or design reuse.

Simplified ease of use: In history-based modeling systems it is necessary to know not only the commands, but the sequence of such commands to complete a workable model. Users are often trained on the myriad of commands. Understanding the required sequences comes only after using the system and explains why experience is so important.

8. REFERENCES

[1]. Weihua Kuang," Development of Parametric Part Family System Based on UG and Excel Parameter Form", 2009 IEEE

[2]. Zhang Dongmin, Hu Dachao, Tang Tingxiao," A Model for Components Library Based on Multilevel Assemblies and Parts Family", 2008 IEEE

[3]. Junmei Xi, Gaohua Liao," CAD System for Plastic Profile Extrusion Die Based on UG**",** 2009 IEEE.

[4]. Fang Feng, Hui Pan, Guojun Hu, "Pro/E Based Parametric Design of Spur Gears", Advanced Materials Research Vols.201-203 (2011) pp 790-794