Automated Approach for Designing Components using Generic and Unitized Parametric Sketches

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ABSTRACT

All Major mechanical CAD software products have provided application programming interfaces (APIs) and environments for users to develop customizations and plug-ins to meet their specific requirements. Automation of design methodologies and processes using the available APIs will result in improved productivity of the designers. Typically, during automation of designing a component, one algorithm for a particular shape of the component is implemented in automation programs. In this paper, an automation approach for solid construction using Completely Constrained Generic and Unitized sketches is discussed, that will enable us to implement one algorithm for a family of shapes. Both the types of sketches, i.e., Generic and Unitized sketches are completely constrained and parametric in nature. Generic sketches are those sketches that are constructed to result in more than one shape by making some minor modifications. The required modifications are explained in detail in this paper. These Generic sketches can be either closed or open. Unitized sketches are special cases of Generic sketches that are open. The Unitized sketches cannot be independently used for solid construction. A combination of these Unitized sketches in a sequence can result in a closed profile that can be used for solid constructions. The closed Generic sketches can be independently used for solid constructions. The change in shape of both types of sketches is with respect to the local reference system and the sketches can be positioned in 3d space by defining constraints with respect to the global reference system. This approach can be effectively used in design automation of components that has finite number of shape variations which are driven by design rules. These concepts have been demonstrated in the design of some aircraft structural components.

1   INTRODUCTION

All leading mechanical CAD software products[2],[3] have provided application programming interfaces (APIs) and environments for users to develop customizations and plug-ins to meet their specific requirements. Automation of a specific design process of a component using the APIs and

programming environments, will shorten the component construction time and improve the productivity of the designers. However, automation is possible only if the design methodology and the process is clearly defined, APIs are available and programming possible. Typically, construction of component parts that have standard shapes but vary only in size (described as family by Monedero [4]) can be automated through a single algorithm and can be used to construct all parts of the component of different sizes. In case of components that have variations in their shapes, the algorithm will become more and more complex and such a program could be difficult to maintain. The algorithm will be still more complex for constructing wireframe geometry programmatically for a component with varying shapes, since a lot of constraints needs to be applied across different elements of the wireframes programmatically and also algorithms for all different shapes needs to be programmed. Such algorithms are difficult to program, would take a lot of effort and also have high probability of being error prone. PLM World User newsletter[1], lists one such similar problem where initially parametric template part files were being used for design automation. In this method it was required that for every part with a unique topology had a unique template part and programming such automation was cumbersome. A solution using UDFs and Knowledge Fusion in Unigraphics[2] is also mentioned for feature topology variations and for wireframes it is mentioned to create the geometry programmatically without using the templates.

Wang, et al [8] have used parametric sketch template in the automation of design of liquid food packaging machinery. Li, et al [6] [7] have also used parametric sketch templates in constructing and optimizing the construction of profiles in reverse engineering. Li, et al [6] views that sketch template design is one of the paradigms that are explored in parametric modeling. The parametric design (modeling) is defined as a process of designing in environment where design variations are effortless, thus replacing singularity with multiplicity in the design process. Parametric design is done with the aid of Parametric Models. A Parametric Model is a computer representation of a design constructed with geometrical entities that have attributes (properties) that are fixed and others can vary. The variable attributes are parameters and the fixed attributes are said to be constraints [10].

Thus the use of parametric template, particularly parametric sketch template, is one of the useful and beneficial methods that can be used to solve shape variability efficiently. The parametric sketch will address the shape variability issue and the template approach will aid in the efficiency of the overall approach.

In this paper, a parametric sketch template based algorithm is discussed which would provide solution to the wireframe shape variation problem. The wireframes created from this algorithm can be used further to construct detailed part 3d geometries.

2   PROBLEM STATEMENT

The problem statement for this work is to arrive at an automation algorithm / approach that will enable the creation of a family of profiles through a single, modular and efficient algorithm.

3   APPROACH

Following definitions and concepts are important in understanding the approach. Though they are widely understood definitions and concepts, it is stated here for explicitness.

- **Sketch** - A collection of geometry (lines, points and arcs) coupled with relationships (parameters, constraints, equations, dimensions and construction geometry laid out in a 2d format [5][8].
- **Constraints** – The notion of constraints implies that the notion of Degree Of Freedom (DOF) [4]. Constraints are two types, geometric constraints (parallelism, perpendicularity, coincidence tangency etc)) and physical or engineering constraints (formulas, conditional relationships) [4]. Geometric constraints are further classified into intrinsic and extrinsic [9]. Intrinsic constraints are constraints on single geometry elements like horizontal, vertical, fixed etc. Extrinsic are constraints between two geometry elements like tangency, coincidence parallelism etc.

- **Under/Over/Completely constrained sketch** - An under constrained sketch means the DOF of the geometric elements is greater than zero which means any parameter value change cannot be resolved into a single consistent solution. An over constraint means it cannot be resolved because there is contradiction somewhere. Completely Constrained sketch means that the DOF is zero and any parameter value change will always result in a consistent solution.

In this discussion there are some new terminologies used. The definitions and related illustrations are given below

- **Generalized sketch** – These sketches are those completely constrained parametric sketches which can be used to arrive at different shapes by some simple modifications to the sketch entities (dimensional / geometric constraints, properties of geometric elements etc). These sketches can be both open and closed. In Fig 1 the closed sketch on the left is a generic sketch since it can give rise to four different shapes as shown on the right.
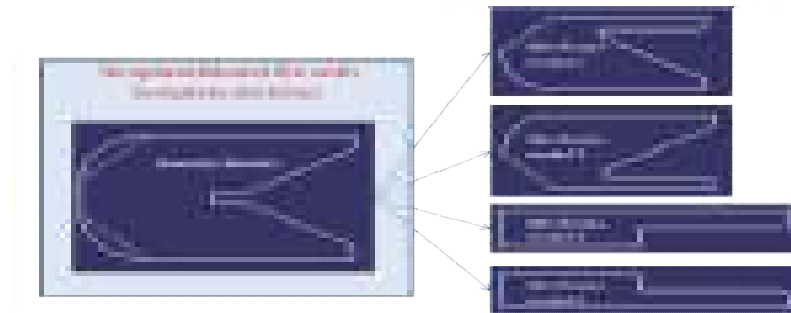

Fig. 1: An illustration of a Generalized Sketch.

- **Unitized sketch** – These sketches are special cases of open generic sketches that form a unit or an elementary shape of a larger closed loop profile. One or more of these unit sketches can be combined in a specific order to realize the final shape of the component. This is illustrated in Fig 2. Two unit sketches at the top in Fig 2 can result in a component shape as shown in below.

Fig. 2: An illustration of Unitized sketch.

The process that could be followed to implement the sketch based approach is as illustrated in Fig. 3 having three major steps.
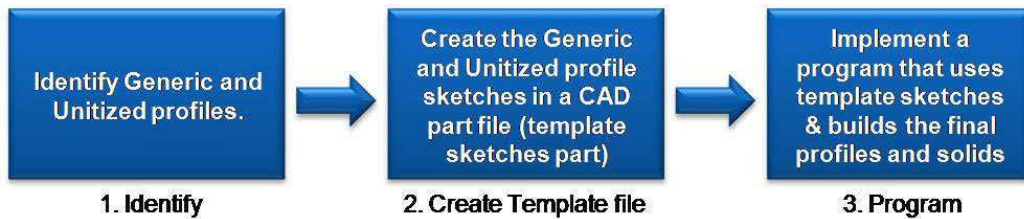
Fig. 3: Process for Sketch based Approach.

3.1 Identify

The objective of this step is to identify the generic and unit sketches that can be used to construct the component solids. In this step, the complete set of component family should be studied and separate out all the profiles that are required to construct each of the solids. The separated profiles should be compared for similarities, differences and partial repeatability in shape and then arrive at generic and unit sketches.

3.1.1 Identifying Generic Sketches

Similar profiles should be analyzed for differences to see if the differences can be added to the similarity of the profiles and arrive at a new generic sketch. The new generic sketch should be parametric and by some simple modifications to the sketch entities, should be able to realize all the shapes that were used in arriving at the generic sketch. It is quite possible that the generic sketch can look completely different from the individual profiles that it is supposed to generate. As seen in Fig 2, the individual profiles on the right hand side of the figure look completely different from the generic sketch from which they could be generated.

3.1.2 Identifying Unit Sketches

The separated profiles should also be analyzed for repeatability of any sub-shapes. All sub-shapes that are repeated can become unit sketches. Similar sub-shapes can also be analyzed further for differences and can be generalized using guidelines given in the previous section. Since Unit sketches are always a sub-shape, it will always be an open profile and it cannot be used independently for construction of solids. The units need to be combined to form closed profiles and the rules of how these units are combined should also exist. Only such units where the rules exist on how to combine them should be identified as Unit sketch.

3.2 Create Template File

The objective of this step is to create a template CAD part file, that contains the Generic and Unit sketches that will be used by the automation program to create specific profiles, which in turn will be used for component solid construction. There are some considerations that need to be considered while constructing the generic and unit sketches. The considerations are as given below.

3.2.1 Considerations to Position the Sketches

The generic and unit sketches are only template sketches, created at any 3d position and it needs to be positioned and oriented in 3d space by the automation program when creating the component profiles. Hence, we need to have a minimum of 3 offset constraints that will at a minimum allow us to position the sketch in 3d space. One of the offset constraints is the offset constraint of the sketch plane from the global reference. The other two offset constraints should allow us to move the entire sketch in the other two directions. We can consider these directions as intra-sketch directions. Two methods were identified for creating position mechanisms for the intra-sketch directions. One of the methods is using construction lines in the sketches and the other is to use the offset Planes, specifically created for positioning

Construction line method: In this method a construction line is created in the sketch and offset constraint is added to this construction line with the external reference. All other constraints are

added with respect to this construction line so that on changing the value of the offset constraint of the construction line entire sketch can be positioned. The primary concern with this method is in some of the CAD softwares available in the market, the offset values cannot be set to negative values programmatically[3], thereby limiting us to position only in the direction in which the offset constraint was created. However this limitation can be overcome by one of the following methods.

Sketch Copies method: In this method two copies of the same sketch are constructed and in one copy of the sketch the construction line has offset constraint on one side of the global reference and in the other the offset constraint is on the other side of the global reference. For positive positioning one copy is used and for negative positioning the other copy is used. This is illustrated in Fig 4. In this method the programming for positioning the sketch becomes simpler but in the template part two copies of sketches for each generic or unit sketches have to be constructed.
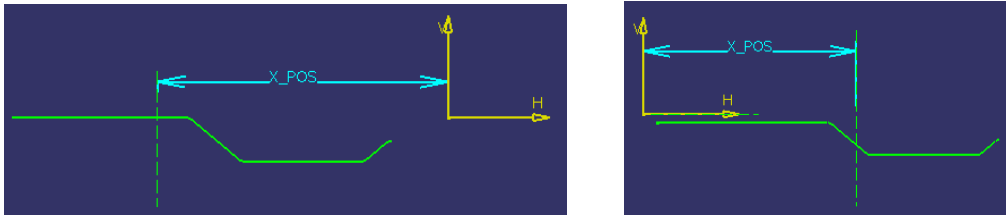

Fig. 4: Positioning Copy sketches (i) Negative position copy, and (ii) Positive position copy.

Parallel Construction Line method: In this method two parallel construction lines; one on each side of the global reference with offset constraint from the global reference is constructed. One of the construction lines is a dummy construction line and is created only for addressing direction issue and the other is the real construction line used for constraining other geometry of the sketch. A reference offset constraint is also added between the real and dummy construction lines. This is illustrated for a simple profile in Fig 5. The original sketch is as shown in Fig 5 (i). When the sketch is to be positioned in the negative direction, then the following steps, which is also illustrated in Fig 5 (ii) through Fig 5 (vii), is performed.

1. Convert the offset constraint of the real construction line from the global reference to reference offset constraint. This is shown in Fig 5 (ii), where Y1 is made reference offset constraint.
2. Convert the reference offset constraint between the real and dummy construction line to offset constraint. This is shown in Fig 5 (iii), where Y1_Dir is made offset constraint.
3. Assign zero value to the offset constraint between the real and dummy construction line. This is shown in Fig 5 (iv), where Y1_Dir is assigned a value of zero.
4. Convert the offset constraint between real and dummy construction line back to reference offset constraint. This is shown in Fig 5 (v), where Y1_Dir is made to reference offset constraint.
5. Convert the reference offset constraint between the real construction line and the global reference back to offset constraint. This is shown in Fig 5 (vi), where Y1 is made offset constraint.
6. Assign the absolute value of the negative offset to the real construction line offset constraint. This is shown in Fig 5 (vii), where Y1 is assigned an absolute value of negative offset.

   In this method, programming will become a little complex but no copies of the sketches are to be created in the template part file.
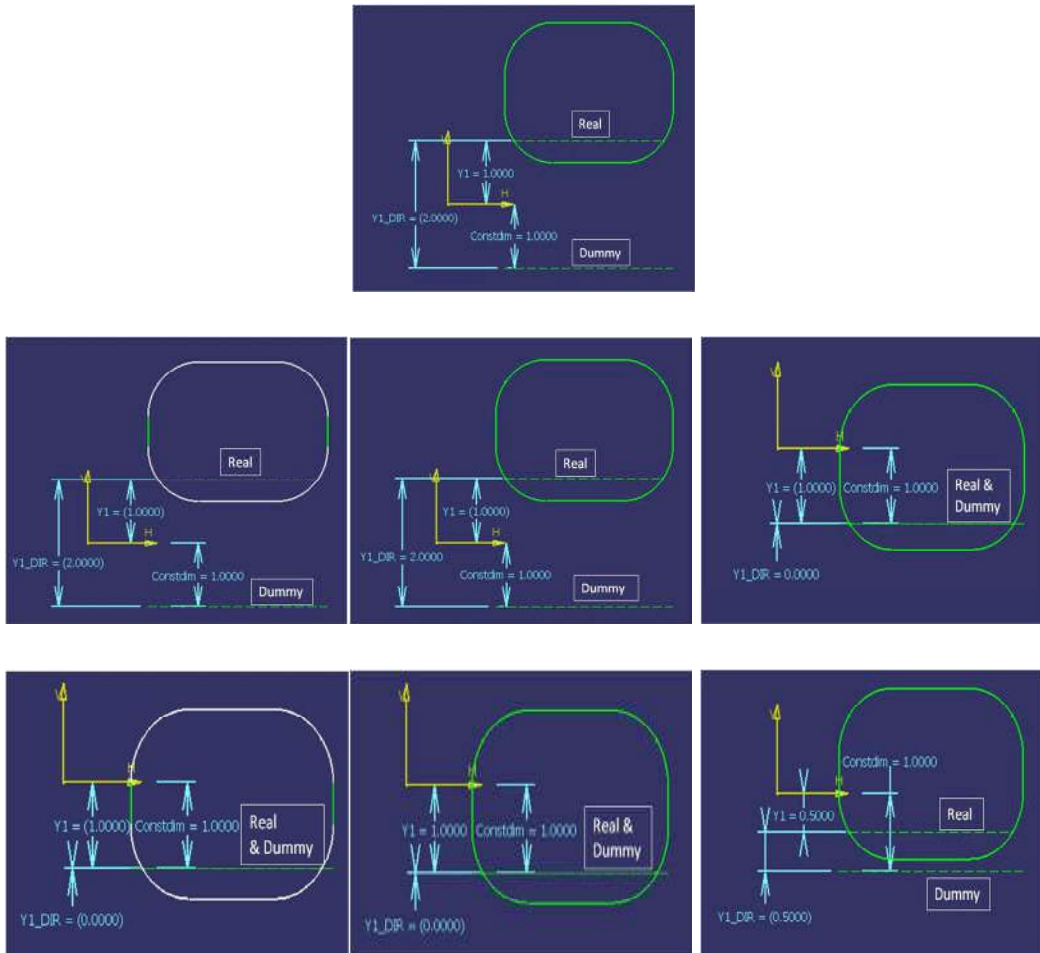
Fig. 5: Steps for positioning sketch in negative direction using construction line method (i) Original Sketch, (ii) Step 1, (iii) Step 2, (iv) Step 3, (v) Step4, (vi) Step 5, and (vii) Step 6.

The construction line method of positioning is particularly useful when the same template sketch is to be used to create multiple profiles in the same part file, since copies of the template sketch can be made and positioned easily by modifying the positioning mechanisms built inside the sketches.

Offset Plane method: The second method to position the sketch in the intra-sketch directions is to construct offset planes from global planes and constraint the sketch geometry with respect to the offset planes. The sketch can be positioned in 3d space by modifying the offset values of the planes. There might be not be a negative positioning issue in this method since in most CAD softwares[3], assigning negative values to plane offsets is allowed. The limitation of this method is that the template sketch cannot be straight away used to created more than one profile in the same part file, since there can be only one set of planes that define the position. Though this limitation can be overcome through some complex programming of creating offset planes for each shape and reattaching the sketch constraints to the new offset planes, this workaround might result in slower runtime performance of the program and is also very complex to program.

Apart from the above mentioned three basic poisoning constraints, depending on the need, angle constraints can also be added to orient the sketch either by using a construction line method or the offset plane method.

3.2.2   *Considerations in Construction of Generic Sketches*

The main consideration that is to be considered while constructing the generic sketches is to build mechanisms into the sketch that will allow us to create many shapes from it. Some of the mechanisms that can be used are given below.

Assigning one or more parameters / dimensional constraints value to zero

Using this mechanism, the geometries associated with the parameter / dimensional constraint is diminished, thereby resulting in a new shape. Fig 6 illustrates this mechanism.
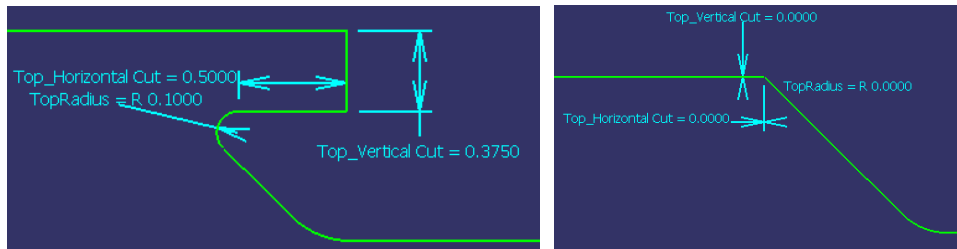
Fig. 6: Assigning one or more parameter / dimensional constraint value to zero (i)   Profile with non-zero dimensions, and (ii) Profile with zero dimensions.

Converting Solid curve(s)/line(s) to construction curve(s)/line(s) and vice versa: Using this mechanism, the geometries that are made solid become part of the profile and we can replace one wireframe with another wireframe in the profile. Fig 7 illustrates this mechanism.
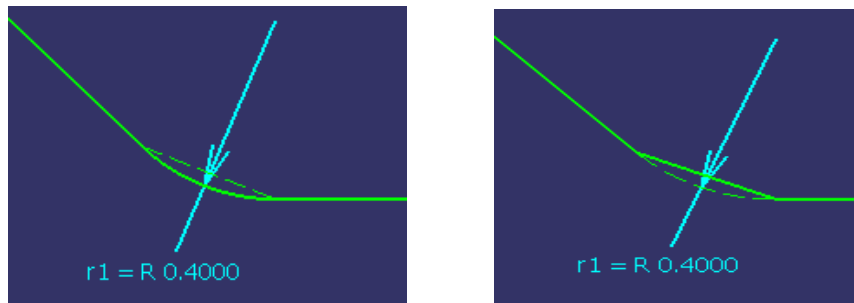

Fig. 7: Converting Solid curve(s)/line(s) to construction curve(s)/line(s) and vice versa (i) Profile with fillet, and (ii) Profile with chamfer.

Converting dimension(s) to reference dimension(s) and vice versa: Using this mechanism, the driving dimensional constraints can be changed and the new dimensional constraints can be used to create new shapes.

Removing or adding or modifying one or more constraints: Using this mechanism we are programmatically modifying the constraints of the sketch and thereby using the new constraints to create new shapes.

Arriving at new shape using the "converting dimension to reference dimension" and "adding new constraint" mechanisms is illustrated in Fig 8. In Fig 8 the following steps are performed to arrive at a new shape.

1. The original sketch is as shown in Fig 8 (i)
2. Convert dimension Distx to reference dimension as shown in Fig 8 (ii)

3. Set the radius ArcR value to zero to remove the arc Arc1 as shown in Fig 8 (iii)
4. Add a new tangency constraint between the vertical line L1 and arc Arc2 as shown in Fig 8 (iv)
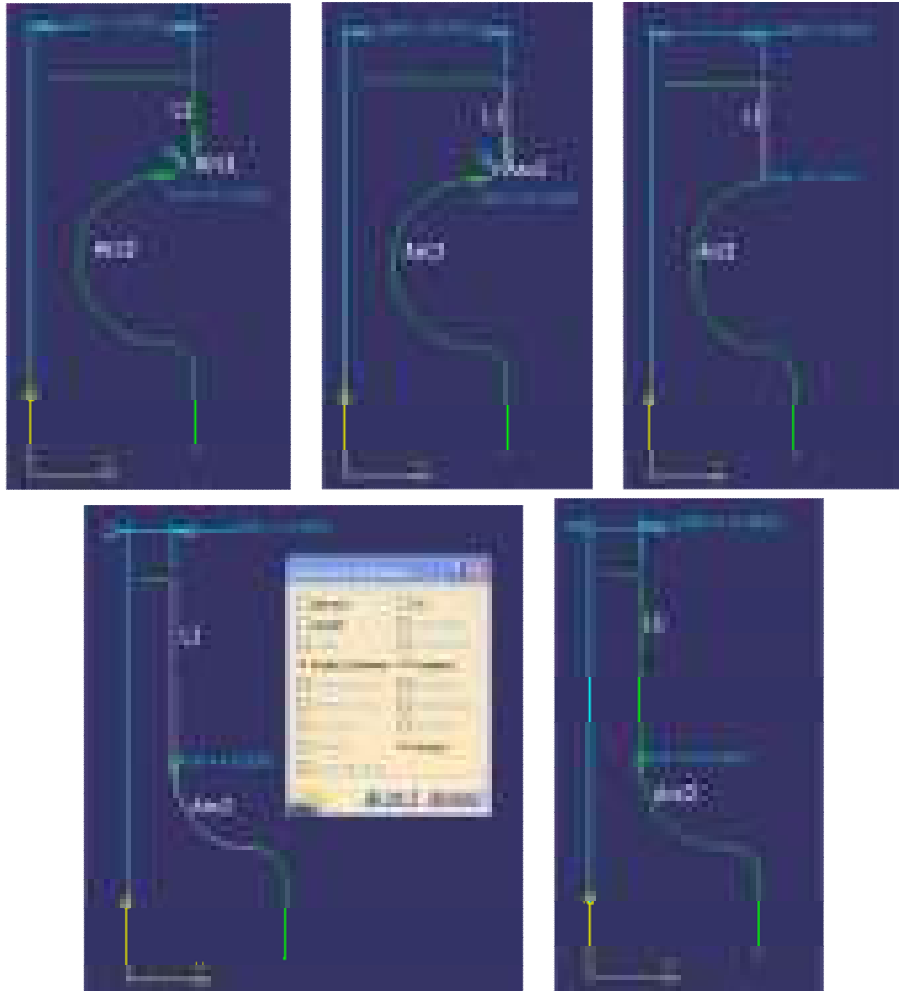5. The final shape is as shown in Fig 8 (v)

Fig. 8: Converting dimension(s) to reference dimension(s) and Removing or adding or modifying constraint(s) (i) Step 1 (ii) Step 2 (iii) Step 3 (iv) Step 4 (v) Step 5.

Considerations in construction of Unit Sketches: Unit sketches are special cases of generic sketches, whose profiles are open and are used to construct a part of the component profile. Hence, all considerations that are described for generic sketches are to be taken into consideration while constructing a unit sketch also. An additional consideration that needs to be considered is the mechanism to make sure that the subsequent units are connected properly at the intersection. One method that was identified is the use of reference dimensions at one end of the sketch. These reference dimensions are such that the values can be directly read and could be used to set the position of nearest end of the adjacent sketch. This is illustrated in Fig 9. To match the end conditions, YPOS1 value in Unit sketch 1, which is a reference dimension, is assigned to YPOS1 value

in Unit sketch 2 and YPOS4 value in Unit sketch 2, which is a reference dimension, is assigned to YPOS4 value in Unit sketch 1
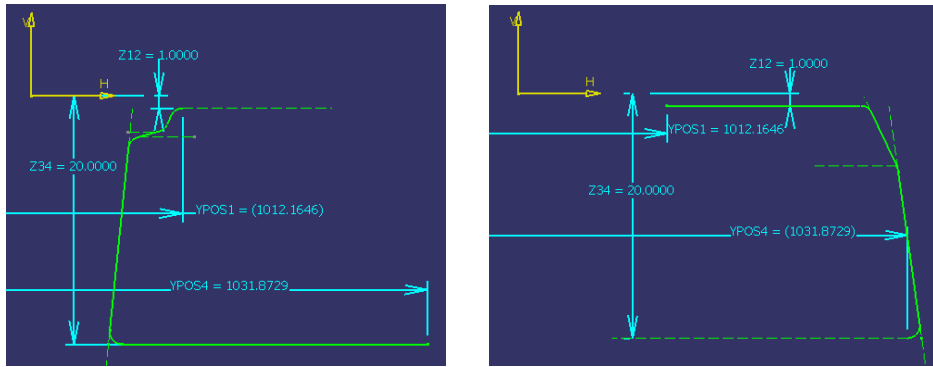

Fig. 9: Matching End positions in subsequent unit sketches (i) Unit sketch 1, and (ii) Unit sketch 2.

3.3 Program

The objective of this step is to implement a program application that will have the engineering design rules embedded in it and will use the template file sketches to create profiles and solids for the respective parts. The program would automate the mechanisms explained in the previous section to create, position and may be orient the respective profile shape for the respective part. The high level application interaction is as illustrated in Fig 10. An application using this approach has been developed in CATIA V5 environment using its VB API [3].
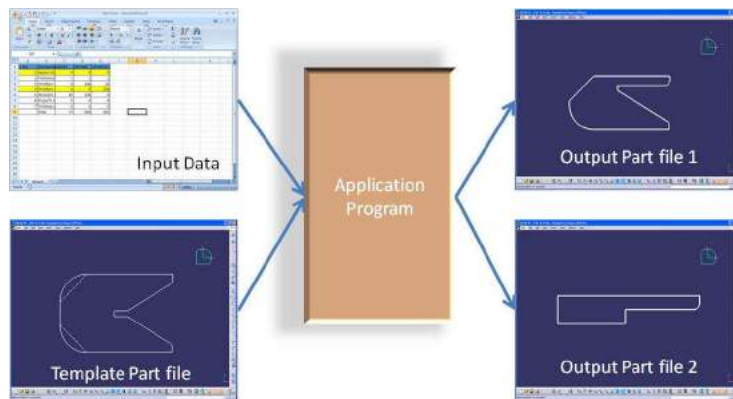
Fig. 10: High Level Application interaction.

4 ILLUSTRATION

The approach is illustrated through a design of a door key. Door keys come in various shapes and sizes. The key heads have various shapes and the teeth profile is unique for every key. The teeth profile is basically a permutation and combination of some basic shapes like the "V", "inverted V", "U", "inverted U" and the likes. The permutation and combination of these shapes is decided by the design of the lock for which the key is made. One of the ways to create a CAD representation of the key is to

create an extruded solid of the base key profile and then subtract the teeth solid created from a teeth profile as shown in Fig 11.
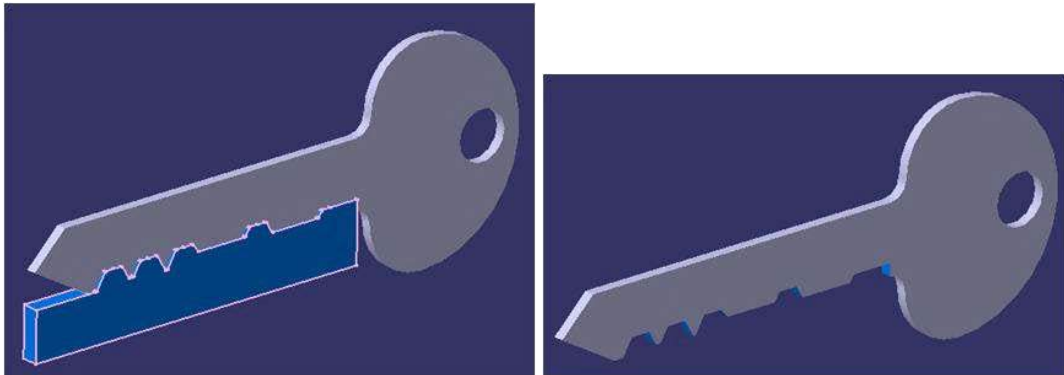

Fig. 11: Boolean subtraction of solids to model the final key geometry.

In this illustration the base profile is constructed from a generic sketch. The generic sketch can be used to created base profiles with different head shapes. In Fig 12, the generic sketch is shown to create four different base key profiles. Fig 12 (a) is the generic sketch, which is also one of the designs. The profile in Fig 12 (b) is realized by converting some of the construction geometry to non-construction geometry and vice versa. Fig 12 (c) is realized from Fig 12 (b) by setting the value of R2 equal to w1+R7+R3. Fig 12 (d) is realized from Fig 12 (c) by converting some non-construction geometry to construction and vice versa.
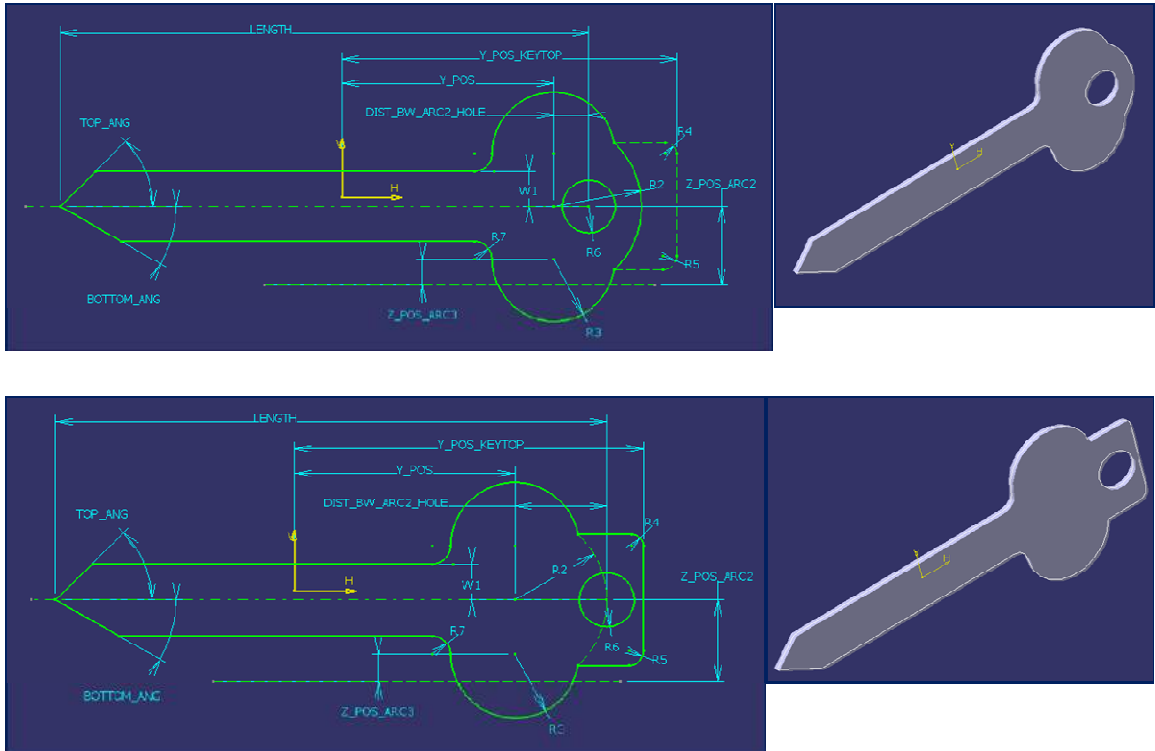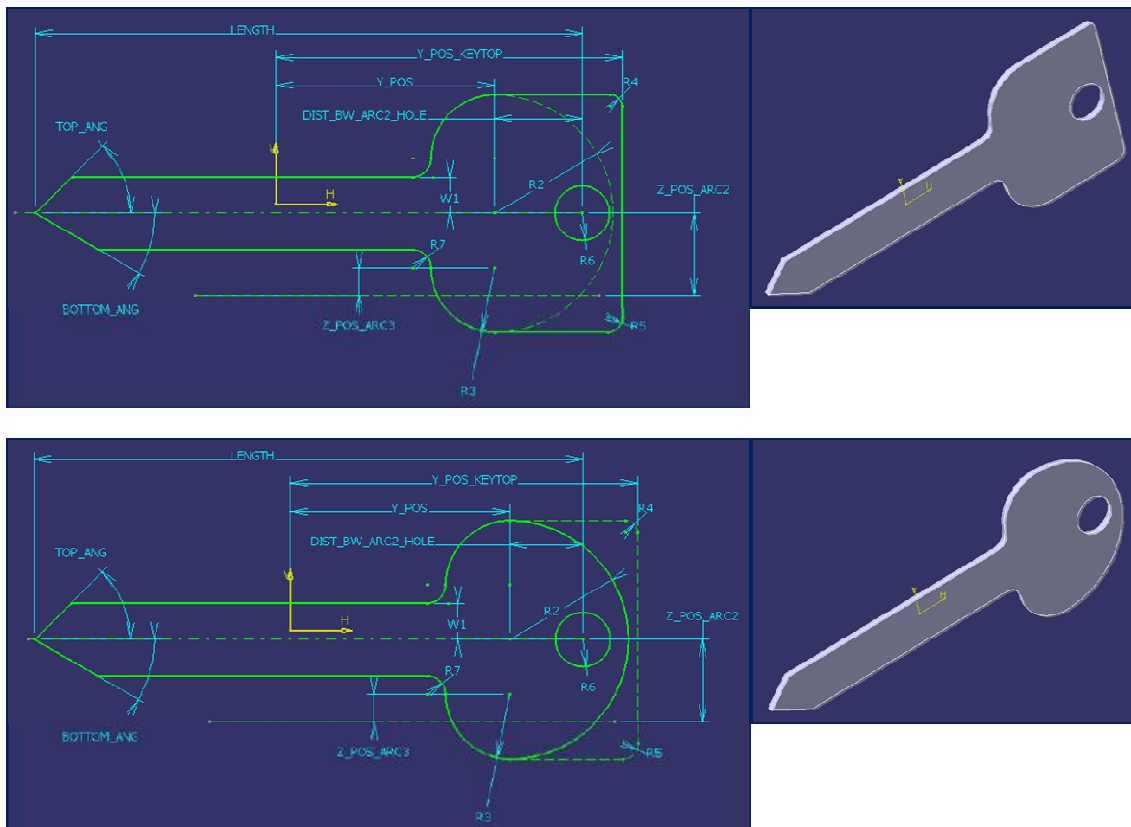
Fig. 12: Illustration of Generic Sketch (a) Generic profile and corresponding base key solid (b) Profile No 2 and corresponding key base solid (c) Profile No 3 and corresponding key base solid (d) Profile No 4 and corresponding key base solid.

Unit Sketches are used to construct the teeth profile. This is illustrated in Fig 13. Fig 13(a) - (b) are the unit sketches. Fig 13 (c) is the required teeth profile. It is made up of six unit shapes. The details of the six unit shapes are also given in Fig 13 (d) – (i). Sketch in Fig 13 (b) is unit sketch used for closing the key profile. Firstly sketch in Fig 13 (i) is realized by using Unit sketch 2 [Fig 13 (b)] by setting the values of START_Y_POS, START_Z_POS, END_Y_POS and END_Z_POS. These values are obtained by the design calculations of the key teeth. Then, sketch in Fig 13 (c) is constructed by using Unit sketch 1 [Fig 13 (a)]. The start is positioned by setting the values of START_Y_POS and START_Z_POS equal to START_Y_POS and START_Z_POS of the sketch in Fig 13 (i). To realize the required shape R1, R2 and ANG1 are set to zero and the value of BOTTOM_Y_POS is set equal to STARY_Y_POS. The other parameters' values are based on the design calculations. The sketch in Fig 13 (d) is realized by using unit sketch 1 and by matching the start position with that of sketch in Fig 13 (c) and setting other parameters based on design calculations. For sketch in Fig 13 (e) again unit sketch 1 is used and the start is again matched with the end of the preceding sketch. The shape is realized by setting value of BOTLINE_LENGTH to zero. Other parameters are set based on design calculations. Similarly sketches in Fig 13 (f) – (g) is realized using unit sketch 1. For sketch in Fig 13 (g) additionally end positions are matched with that end positions of the sketch in Fig 13 (i). Thus we have created a closed loop teeth profile for this particular key. Different keys will have different numbers of teeth and the order of the shapes will also differ. Using only the above mentioned two unit sketches, teeth profiles with any number of teeth and also with any order of shape can be constructed.
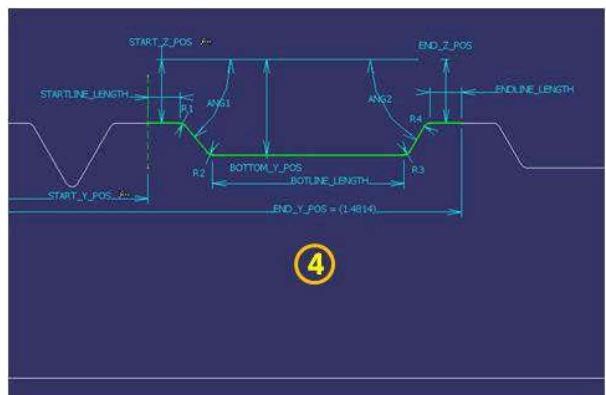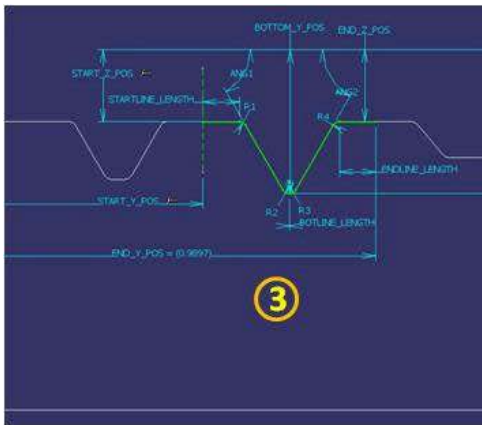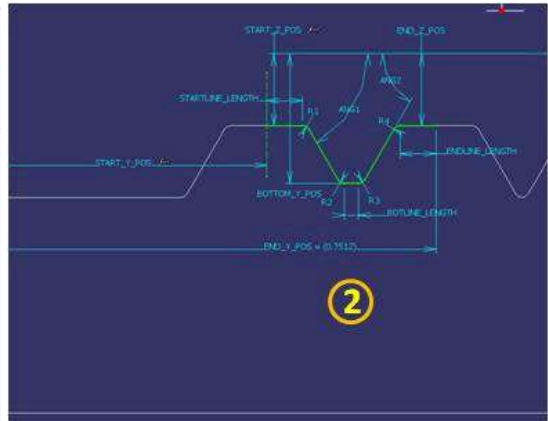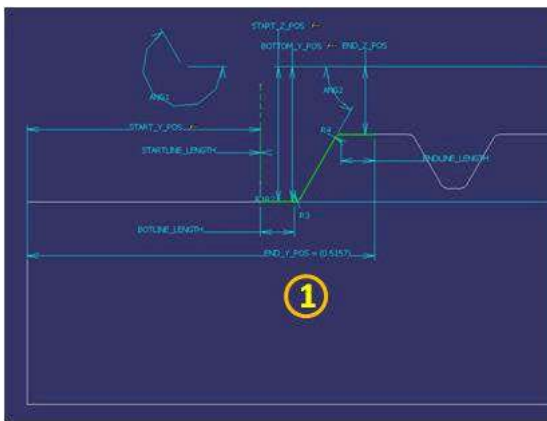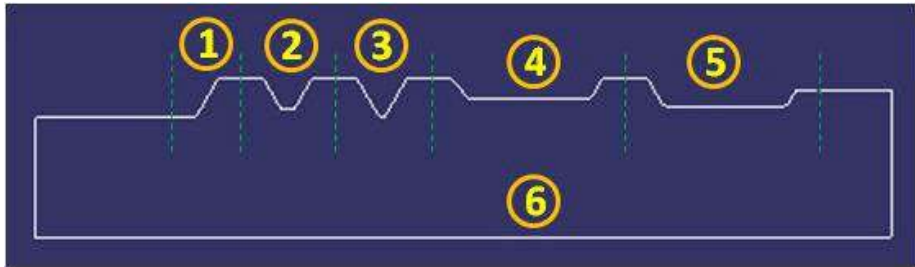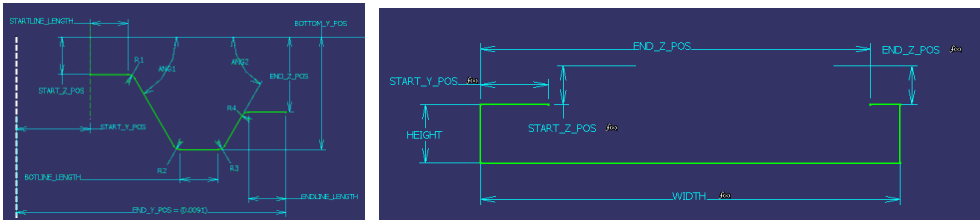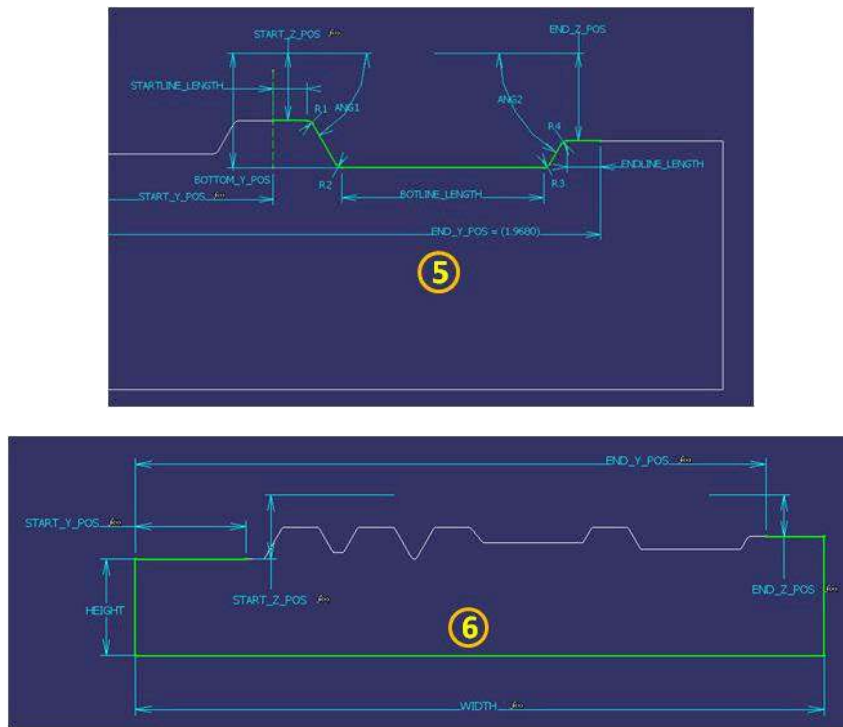
Fig. 13: Illustration of Unit Sketches (a) Unit sketch 1 (b) Unit sketch 2 (c) Key teeth profile (d) Unit 1 of key teeth profile (e) Unit 2 of key teeth profile (f) Unit 3 of key teeth profile (g) Unit 4 of key teeth profile (h) Unit 5 of key teeth profile (i) Unit 6 of key teeth profile.

5   ADVANTAGES

Some of the advantages of this approach are given below

Ability to handle Shape variability: The generic and unitized sketches can help in creating a large number of shapes, sometimes even shapes that are not part of the base design. Using the mechanisms mentioned in the previous section many shapes can be derived from generic sketches and also using permutation and combination variations of the sequence and the type of unit sketches, many shapes can be derived.

Modularity: The geometry related elements of the program is separated and most of it are embedded in the template part file. The application program would primarily contain the design engineering rules in it. This makes the entire structure to be modular. Further the template file can be made modular using the construction tree capabilities of the CAD software and the program also can be implemented in a modular way using the Object Oriented Application Design concepts.

Extendibility: The modularity of the approach to some extent is responsible for easy extendibility of the application. New profile sketches can be added to the template files, whenever new family of shapes are encountered and new code related to the new shapes have to be implemented.

Better Performance: Since this approach does not use some runtime performance wise expensive CAD operations, the runtime performance of applications developed using this method is better. In one of the modules which were initially implemented using expensive operations like wireframe intersection, trim etc, on re-implementing with this approach, improved the runtime performance of the module by a minimum of 80%. The details are given in Table 1

Part No	No Of features	Runtime with old approach (mins)	Runtime with new approach (mins)	Difference (mins)	% Improvement
Part 1	13	6	1	5	83.33%
Part 2	14	8	1	7	87.50%
Part 3	18	13	2	11	84.62%
Part 4	17	10	2	8	80.00%
Part 5	14	9	1	8	88.89%
Part 6	16	8	1	7	87.50%

Tab. 1: Quantification of Performance Improvement.

Shorter Application Development time: The effort required to creating sketches and constraining the sketches manually is less when compared to the effort in implementing a program that would automatically create the sketch, even for a simpler sketch. Hence the application development time is reduced when using this approach since a part of the programming effort would be shifted to sketch creation effort.

6   LIMITATIONS

Some of the limitations of the approach are given below

Effort required to study of shapes: Effort required to study the shapes can be huge. If sufficient time is not spent on the study, there could be a possibility that the derived generic and unit sketches may have limitations and could be difficult to modify / extend further, when the missing parts are discovered later.

Too Many parameters in a sketch: If the generic or unit sketch contains too many parameters as a result of combining a large number of shapes, it can become complicated to maintain and use such a sketch.

Too many modifications required to realize a shape: If too many modifications are required to realize a shape from a generic or unit sketch, the programming can become complicated and difficult to maintain.

7   CONCLUSION

The approach described in this paper is applicable only when there is a visibility of the shapes of the component family in the initial phases of the design. The shapes should be such that generic and unit sketches can be derived. Judicious decision on the applicability of the approach is to be made based on effort involved in studying the shapes versus the resultant benefits and then the approach needs to be adopted. Also while adopting the approach the limitations mentioned in section 5 should be kept in mind. Overall the approach has huge benefits and is applicable in design of most of the structural components. The approach has been implemented in the design of one of the structural component of an aircraft and had helped in realizing the benefits mentioned in section 4.

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