

## **RULES OF DEFINING SKETCHES IN PRO/ENGINEER WILDFIRE / CREO / CREO PARAMETRIC**

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**Abstract:** Parametric modelling in CAD applications involves performing the next steps: defining a generating sketch; sketch movement controlled by a trajectory imposed by the command launched in execution. In ProEngineer Wildfire / Creo 0 / Creo 1 CAD application, under certain conditions, when we make a symmetrical copy of a form of revolution it is placed asymmetrical to the selected reference entity without any system error. The placing error occurs as a result of how it is defined the generating sketch. This paper investigated the causes of error after which are drawn conclusions on how to define the sketches so that such errors to be avoided. The proposed method of defining the sketches is generally valid and it is recommended to be applied in defining all forms in this CAD applications.

### **1. INTRODUCTION**

In instructor guide for ProEngineer/Creo is said „When creating models with Pro/ENGINEER Wildfire, it is critical that you capture the design intent of the model. Design intent ensures predictable results when a model is modified” ALSO in ProEngineer documentation [4], [5] the steps necessary to define generating sketch of the features based on sketches are:

- specify the sketch setup;
- select additional sketch references that you intend to dimension from or snap to with sketch geometry;
- sketch the geometry.

No restrictions or recommendations are made on references selecting process [4], [5]. By default in orienting process the system selects (in most cases) 2 orthogonal references, i.e. geometric projection of surfaces or datum planes that are used in guidance on sketching plane

From the point of view of the completion of the generating section is indicate the next steps:

- define a sketch similar to the desired profile (without dimensioning it);
- detail the sketch (dimensions and constraints).

Profile dimensioning is done by dimensions and/or constraints that must reflect the design intent (the fulfilment of the aims of the functional and technological of shape); allows getting predictable results in case of changes), (Kelley, 2009), (Toogood, 2009).

On the basis of the PTC-Parametric Technology Corporation documentation, reference publications and experience, the author (Manolea, 2011) has offered the following suggestions for shape-generating step:

- select and orient the sketching plane. The orientation of the sketch is made with the help of geometric entities of the existing model;
- entities selected in the orientation stage are by default selected as references in sketch. The user can choose additional new references. It is useful to be selected all the dimensioning references of the new feature;
- generate the sketch similar to wanted profile, but without imposing dimensions. In this phase shall be used previously defined references;
- dimension and constrain the sketch. In this stage the system automatically defines as references all the geometric entities that have been used for sketch dimensioning.

In Creo, ProEngineer Wild Fire v5.0, Creo Parametric versions, after shape validation, the system automatically delete the duplicated references and the references contained in other references. The references defined by the system as a result of a dimensioning or a constraining operation are not deleted. The validity of those recommendations has been ruled out (in certain situations) by the results of the MIRROR command use. In certain situations, for the same reason, errors are obtained with BLEND command.

This paper examines these situations and proposes a way of avoiding them (in case of MIRROR command).

The original ideas of the paper are:

- in sketching stage avoid choosing as reference cylindrical surface;
- sketch is positioned in an arbitrary position, then get the correct position by constrains applying.

The application of these ideas leads to getting models that respond to the principal aim of modelling, namely that the created model is easily dimensional and shape modifiable.

The application areas of these ideas is generally available in product modelling using CAD software.

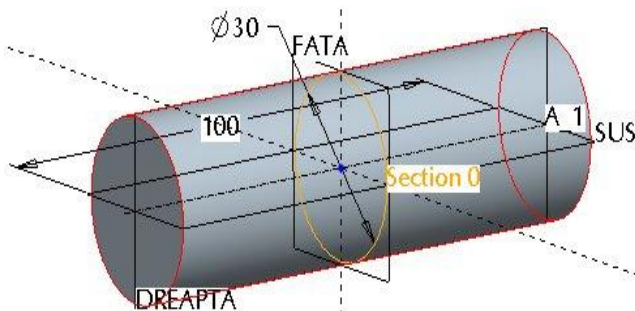
The paper emphasize a problem that appears in the stage of part modelling, and offer a solution to this problem. The solution is generally valid, verified, and we believe that for these versions of the program, further research are no longer needed.

We consider useful to draw attention that in the teaching of CAD software should be granted a major importance to the techniques that will ensure getting a correct model (easily modifiable model that responds to the design intent).

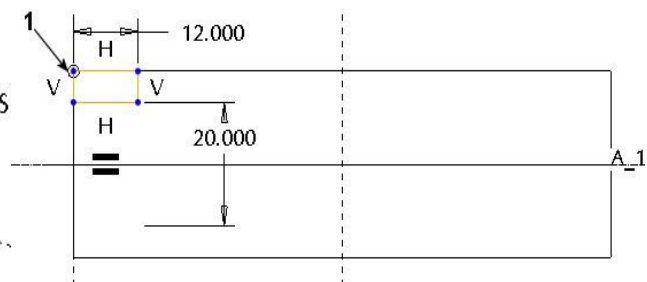
Conclusion of the work provides such a technique, namely the stages to be completed in sketcher environment of the ProEngineer software.

## 2. A SIMPLE MODEL WITH IMPORTANT IMPLICATIONS

Is generated a cylinder by a symmetric extrusion command – figure 1. Can be used the revolve command with similar result.



**Figure1. First feature**



**Figure2. The sketch of the revolved cut**

Using a revolved feature, remove material at the left end of the cylinder – figure 2. The dimensions are not important.

The sketch must be made following the next steps:

- select the DREAPTA datum plane with default references (default orientation) as sketching plane;
- select as revolving axe the projection of the SUS datum plane;
- select as references the cylindrical surface and the left plane end surface of the cylinder;
- sketch a rectangle with the initial point positioned at the intersection point of the

references – pnt. 1 figure 2. It is very important the position of this point;

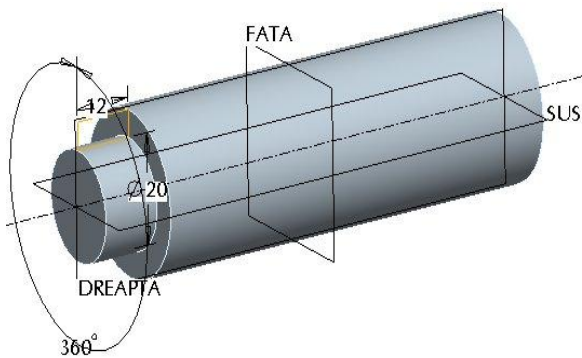
- the dimensions are not important;
- validate the sketch and the feature – figure 3

Mirror the last made feature using as reference datum plane FATA – figure 4.

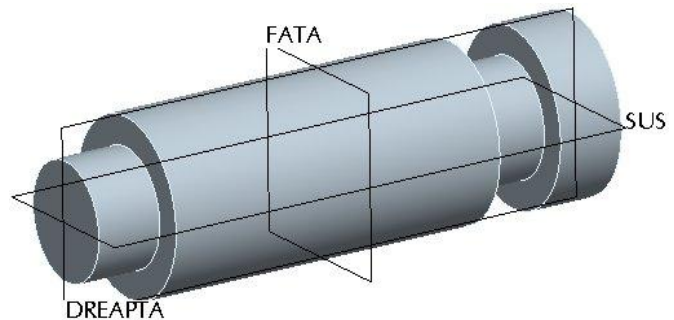
Note that the new feature is not symmetrically placed respecting the selected reference plane (FATA).

Through successive tests were established the following conclusions:

- error is not dependent on version (the same result in Creo/ProEngineer-WildFire5.0 and Creo 1);
- error is not dependent on the profile of the sketch – figure 4, figure 6 – rectangle; figure 5 – semicircle;
- error is not dependent on feature type – figure 4, figure 5 revolved feature; figure 6, figure7 extruded feature;

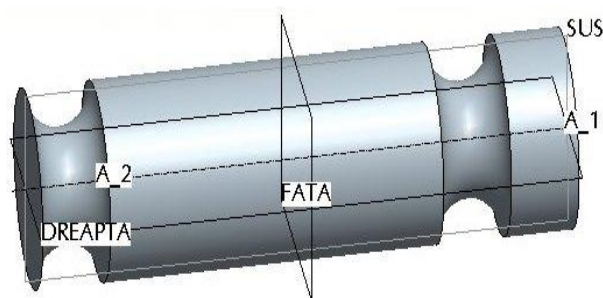


**Figure 3. Revolved cut**

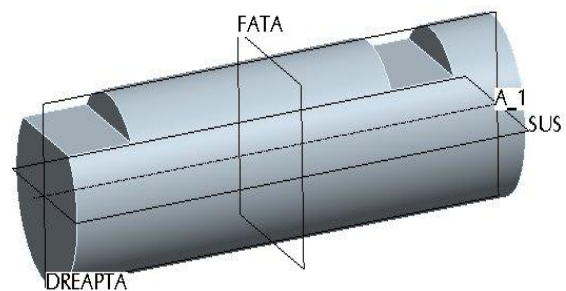


**Figure 4. The resulting model after the mirror command**

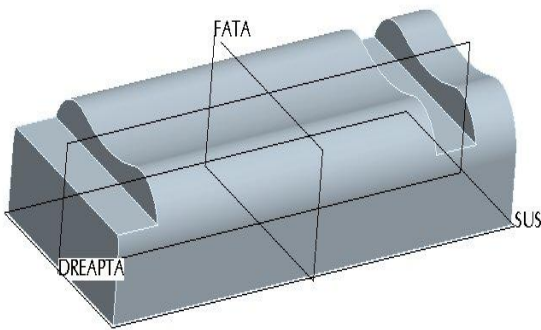
- error occurs only if the parent feature is a revolved one – figure 2, figure 4, figure 5 or if the side of the feature is composed of cylindrical surfaces – figure 6 (section giving rise to the surface is made up of arcs);
- error occurs only if the sketch starts from the point located at the intersection of the two references (the cylindrical surface and flat surface - point 1 in figure 2) defined by the user;
- the error does not occur if the shape is obtained through an outline of spline curves – figure 8.;
- in all cases where the error appears, the system automatically selects as single reference the cylinder surface and remove other references – figure 9 references selected to create the shape, figure 10 references after the form has been completed.



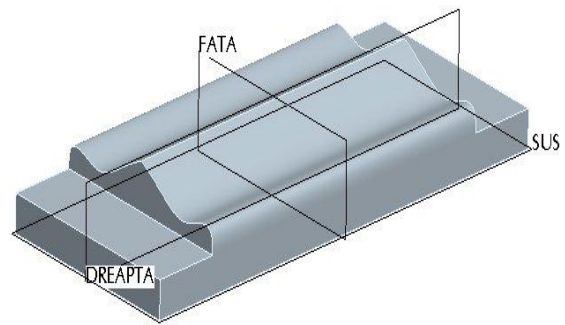
**Figure 5. Revolved semi-circular profile**



**Figure 6. Extruded rectangle profile**

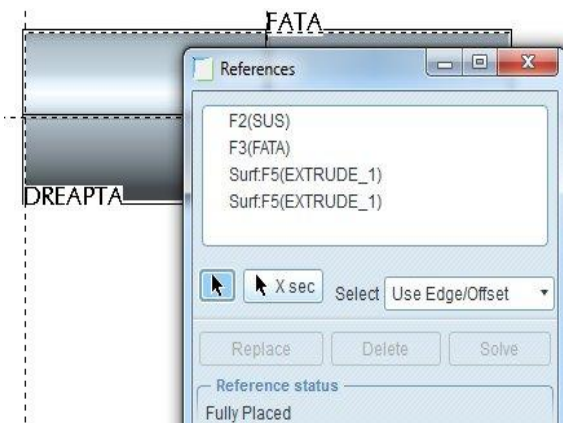


**Figure 7. Extruded rectangle profile. the upper surface of the model is composed from cylindrical surfaces**

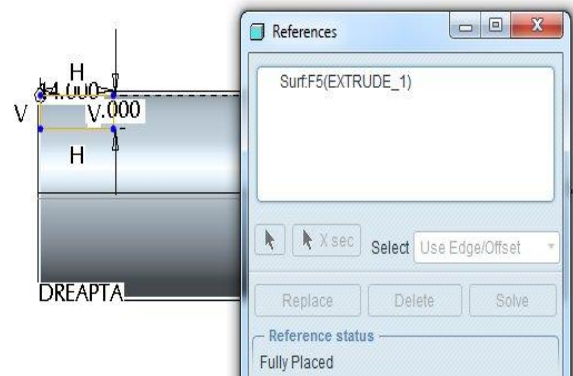


**Figure 8. Extruded spline profile.**

The final conclusion is: if you sketch the profile of the section in an arbitrary position and align the left edge of the rectangle with the flat surface and the upper edge of the rectangle with cylinder surface then the error no longer appears.



**Figure 9. References before the feature has been completed**



**Figure 10. References after the feature has been completed**

### 3. CONCLUSIONS

The error occurs as a result of the automatic selection of the cylindrical surface as the only reference positioning. Pro/ENGINEER has introduced as an "improvement" keeping to a minimum the number of references required for positioning of the sketch (from Pro/ENGINEER WildFire 5.0 version). Not to be confused with references that are defined by the system as a result of dimensioning or as a result of the applied constraints (these references are not removed automatically). The error does not occur in the case of surfaces which are defined by spline curves because the system cannot use these surfaces as references for positioning the sketch.

The error occurs when using the mirror command but can also occur if the copy command is used.

The dangerous nature of this type of error is due to the fact that: it is not detected by the system; it can be observed only by the user (operation difficult in the case of models with tens or hundreds of features). To avoid this kind of errors we recommend completing the following steps to define the sketch:

- select and orient the sketching plan;
- sketch the cross sectional shape of the feature on an approximate position;
- the correct sketch position will be achieved by imposing constraints. Doing so, the

system can no longer remove the references.

We believe that it is not recommended to select additional references, since the system is free to remove them in order to maintain a minimum of positioning references. This „optimization” is the root cause of the occurrence of the error is dealt with in this paper.

#### **4. IMPORTANCE**

We consider important the conclusions of the paper because:

- it prevent the occurrence of errors that appear in the final phase of the modelling;
- the errors are difficult to detect in the case of complicated models with tens or hundreds of features and cannot be solved than modelling the part again.

#### **References:**

1. Kelley D. S., (2009). Pro/ENGINEER Wildfire 4.0, pp. 55-65, McGraw-Hill, New York, ISBN 978-0-07-352266-1
2. Manolea D., (2011). Proiectare constructivă prin Pro/ENGINEER, pp.24-30 , Ed. Universității “Lucian Blaga”, Sibiu, ISBN 978-606-12-0165-5
3. Toogood R., (2009). Pro/ENGINEER WILDFIRE 5.0 Tutorial and MultiMedia CD, pp. 2-1 – 2-14, SDC Publications, Edmonton, ISBN 978-1-58503-535-9
4. Introduction to Creo Elements/Pro 5.0 Available from: <https://precisionlms.ptc.com/app>, Accessed: 12/10/2011
5. Creo elements/Pro Help, Part modelling, Sketcher, References