



Pro/E Tips and Tricks

1. What is the difference between Scheme and Sketch?

Both are used as options to redefine a feature's sketch, but Scheme doesn't allow Modify. Scheme allows only for modifications of the dimensioning scheme, and does not allow deleting the sketched entities.

2. What is KNOW dimension?

KNOW dimensions are dimensions "known" by Pro/E as value which can be realized only between elements that are already existent in the model. They can be added in relations at the sketch level, ex: $sd\# = kd\# / 2$.

3. Why is important to begin a model with three datum planes?

Any Pro/E model can be started by creating 3-ortogonal planes -Default Datum Plane- before modeling any shape.

These planes are created in order to place, dimension, and orient the model. The 3 default datum planes represent the non-solid base of the model. The first solid feature will be placed and referred to these 3 planes. The 3 default planes can be created only once at the start of the model. The majority of design companies use a "start part" that includes these 3 planes, saved views, layers, specific parameters, etc.

The use of these planes is very useful for the following reasons:

- They lead to correct parent-child relationships, eliminating unwanted relations;
- They can be used as plane surfaces in models that don't have such surfaces;
- They are the basis for any model's creation. They can be manipulated with the help of Layers or from Environment Menu.

4. What do REORDER, SUPPRESS, RESUME, and REDEFINE do?

Reorder allows you to reorder the features in Model Tree (changing the regeneration order of the features). The system specifies to the user the position in which the selected features can be reordered.

Suppress allows the temporary suppression of one or several features from active memory (with exclusion from the regeneration list) in order to:

1. design alternatively



2. focus on a special area
3. reduce the regeneration time

Resume is the opposite of Suppress; it displays the suppressed features (with inclusion on the regeneration list).

Redefine allows to the user to redefine an already created feature, following the same steps used in creation of the feature:

1. attributes
2. sketch and reference planes
3. depth options

5. What are SKETCHING plane and PLACEMENT plane?

SKETCHING plane is the plane in which the sketch is realized. A sketching plane can be any hard datum plane, plane face of the model, or you can utilize a datum plane "on the fly".

PLACEMENT plane is the plane, which orients the sketch. This has to be perpendicular on the sketching plane. This can also be any hard datum plane, plane face of the model, or you can utilize a datum plane "on the fly".

6. Which face of a (reference) datum plane will be aligned in the specified direction?

Any plane surface of a model or datum plane can be used as a reference horizontal/vertical plane, with the condition that these are perpendicular on the sketching plane. If you use a datum plane, the orientation of the sketching plane is done by orienting the reference plane with the yellow face (positive side of the plane), parallel with one of the display's edges, by using commands Top, Bottom, Left or Right.

All sketched features (sketch and reference planes) will be oriented by using the yellow face if nothing else is specified. Also when planes are chosen for assemblies constraints, the system chooses the yellow face of the plane, implicitly.

The sketching plane is oriented parallel with the display. The system shows an arrow which indicates the direction of creating a shape.

7. What is a Parent-Child relationship?

Modeling in Pro/E is a cumulative process; this is why some forms will precede others in Model Tree. When you create a new shape, this will refer to shapes modeled previously, and these become "parents" to the new shapes. The



interdependency relation between a newly created shape and the shapes to which it was referred is called "Parent-Child" relationship.

Here are some of the ways to obtain a "Parent-Child" relation: alignments, dimensioning, dependent copying, choosing the sketching or reference planes, patterning, etc.

8. What can you do with REROUTE?

Reroute allows you to change the references that were previously used to define a shape by choosing new references with option Alternate, or by keeping the old references with option Same Ref.

The systems offers 2 possibilities at the reroute-area of a shape, through option "Roll back":

- Yes - the features regenerated after the rerouted shape are inactive.
- No - the features regenerated after the rerouted shape are active.

9. What are the benefits of using INSERT MODE?

It allows the insertion of new features in any area of the model. The system offers 2 options:

Activate - this option suppresses the features beyond the insertion point automatically, allowing the creation of new features.

Cancel - this option allows you to deactivate the insertion mode. All features suppressed with Activate will be regenerated.

10. List three reason for using the SUPPRESS feature.

Suppress represents one of the most powerful methods used in Pro/E to simplify the part, assembly or drawing creation processes. When you suppress a feature or a component in a model or assembly the system temporarily stops regenerating them. Suppress is different from deletion because it is not permanent.

By using suppress you can achieve the following:

- Model display modification: allowing you to concentrate attention on the current work area
- Faster regeneration of the model and shortening of the repaint time
- Alternate design
- Elimination of problems that can appear in regeneration
- Ability to save small dimension files.

11. What is a datum ON THE FLY?

This is a datum created during the creation of a feature and can be used as a sketching, reference plane or planes that determines the depth of a feature. It is



visible in Model Tree only during creation of the feature, but the dimensions that were used for its creation can be subsequently modified with Modify or Redefine for the respective feature.

12. You want to pattern a sketched feature. What is the best method to obtain the angle that will be the pattern incremental dimension?

In order to create a rotational pattern, it is necessary to create an angle dimension to place the leading feature. The angle dimension used at the rotational patterning must be obtained with the help of a Datum On The Fly obtained during the creation of the feature, with the option Make Datum from Setup Plane Menu.

13. What is a reference pattern and what is the condition to generate one?

The reference patterning of a feature (or group of features) can be done if this feature was referred to as the leader of a pattern already created. The reference pattern is done with the same rules that were used to create the previous patterning (parent).

14. What are the layers in Pro/E and why do we use them?

A layer in Pro/E represents a model entities grouping (ex: features, datum planes or axes.) The layers are used to administer the entities in a model, assembly or drawing in Pro/E. Several entities can be associated to a layer, so as to manipulate them together. They can be suppressed if they are solids, and hide or be displayed in an organized fashion if they are non-solids.

The layers help to simplify a model and increase work speed.

The visibility of non-geometric entities can be controlled in a model or in assembly components with the help of layers. Also, unnecessary elements of a drawing (surfaces or datum features) can be hidden.

15. What kind of items can layers manage?

Layers can manage all types of items. After they were created, several items can be associated to layers, such as: features, curves, surfaces, datum planes, text, dimensions, geometric tolerances, etc. An entity can be placed in a layer in the following ways:

- Manually, by selecting the entities individually
- By copying layer information
- By creating layers and associating specific attributes automatically



16. List the view types that exist in Pro/E?

General, Projection, Auxiliary, Detailed, Revolved, Graph, On Flat Ply, Copy & Align
Full view, Half view, Broken view, Partial view
Section, No section, On surface
Scaled, No scale, Perspective
Xsections type: Full, Half, Local, Full & Local, Total Xsec, Area Xsec, Align Xsec, Total Xsec, Unfold Xsec, Total unfold

17. How can you make a parametrical note?

With the help of "&" symbol, placed before the parameter or dimension that we want to bring in that note.

18. What is the X-Trajectory for a VarSecSweep feature?

A Variable Section Sweep feature is created by moving a section across one or more trajectories. The size of the section can vary as it moves across the trajectories.

In order to create a Variable Section Sweep perpendicularly on the Origin Traj, it is necessary to define at least one additional trajectory, called X-Trajectory. The system uses this trajectory to orient the section in the intersection point, across its entire movement along the Origin Trajectory curve.

The flexibility of a model is increased by selecting the datum curves as sweep trajectories, instead of sketching them.

When creating a Variable Section Sweep feature, the X-Trajectory can only intersect the Origin Trajectory in its end points.

19. What is the role of the additional trajectory in a VarSecSweep and how many of these can there be? Can you make a VarSecSweep feature without additional trajectories?

Additional trajectories can be selected or sketched in order to define the features, once the Origin Trajectory and X-Trajectory were defined. The system automatically adds a point at the intersection between the additional trajectories and the sketching plane. Since the system moves the section plane along the trajectories, its intersections with these will represent "known points" and will be used only for dimensioning and / or alignment of the section.

These trajectories are optional. If they are selected and the section is aligned by them, then they will control the shape of the section during its movement.

There are no restrictions as far as the number of additional trajectories used in a VarSecSweep feature, but if they are used, they must intersect the sketching plane. Also, it is not necessary that the additional trajectories have the



same length as the Origin Trajectory, because the system realizes the feature with the length of the shortest trajectory.

20. Can you make a VarSecSweep feature without X-Trajectory?

No. In order to create a VarSecSweep perpendicular on a trajectory - there are necessarily at least 2 trajectories: Origin Trajectory, the one on which the section plane will be perpendicular, and at least another trajectory, named X-Trajectory, which will orient the section while it is moving.

21. What is the difference between Normal to Origin Trajectory and Pivot Direction in a VarSecSweep feature?

While in the case of Normal to Origin Trajectory the movement is made perpendicularly on the Origin Trajectory (the orientation of the section is made based on X-Trajectory) in the case of Pivot Direction it is necessary to establish a reference (datum plane, datum axis or datum curve) so as to the section to remain perpendicular on Origin Trajectory.

In the case of Pivot Direction, if we choose a nonlinear curve as reference, the system will require us to specify a point on this curve, against which it will determine the direction of the curve's tangent and it will determine it's perpendicular against this one.

22. What is the typical parameter for VarSecSweep and how can it be used?

When a VarSecSweep regenerates, the system evaluates an internal parameter named Trajpar. This parameter represents one of the most powerful functions of the features type VarSec Sweep. Its value is between 0, at the beginning of the feature and 1 at its end and represents the percentage of the sweep feature's length.

This parameter can be used in relations who will control the section. We must mention the fact that the trajpar relations can only be used at the sketch level and that the system evaluates the relation only when it regenerates the feature.

The trajpar parameter can be used to lead the surfaces to 0 anywhere along the trajectory, either by using complex relations to lead the section, or by evaluating datum graph features with the help of evalgraph function.

23. What is the Start Part and what are the benefits when you use one?

Start Part is a part that is used at the beginning of each part from a



certain project. Thus, the information contained in the Start Part will be found in each part modeled in the project. Usually, this information is related to: planes and csys default, layers and elements associated by default to these, saved views, measuring units, accuracy, default xsections, default relations etc.

The main advantage of using the Start Part is that in a certain project each part will be started in the same way and the parts keep the information that are found in Start Part. Moreover, you can save time, the Start Part being similar to a template. For the same project it is not necessary to introduce the information contained by Start Part for each part.

24. What does REPLACE from Geom Tools menu do? How many Replaces exist in Pro/E?

REPLACE in the Geom Tools menu allows replacing an entity from a sketch with a new one, while maintaining the parent - child relations held by the previous entity.

Sketcher: Geom Tools-->Replace and Dimension-->Replace
Part: Feature-->Create-->Tweak-->Replace
Assembly: Component-->Adv. Utils-->Replace

25. What does "WARNING: Unattached Feature" mean?

Due to Pro/E parametric nature, any model modification on any level is propagated in the entire assembly. The parametric relations between features, parts and assemblies realize connections known as parent-child relations. If any of them are broken, the regeneration process is interrupted.

The most frequent causes are the ones in which the system can no longer build a feature or assemble a component. This may happen in any stage of making the model: its creation, modification or loading in the work session.

Pro/ENGINEER successfully regenerates the features or components that are placed before the one that poses problems, and when the regeneration process stops, the systems places the user automatically in Resolve Environment.

"WARNING: Unattached Feature" is the warning message given by Pro/E when a feature doesn't have a contact with the model (does not intersect it, it is created in the exterior of the model) due to modifications which occurred before this one in the model, or simply because it was built wrong.

26. When you are in Resolve Mode, how can you use the Backup Model?

When an error interrupts the regeneration process, the error must be solved first in order to maintain the design intent. Then the regeneration may be continued.

Often, the most difficult problem that has to be solved is to establish the



cause that lead to regeneration interruption. If the information supplied by the systems in the diagnosis window is not sufficient, you can use the menu Investigate to find other methods that can help to establish the cause. Backup Model allows investigating the problem on a back-up copy, in the state of the latest successful regeneration. The "problem" feature will not be displayed.

27. When you write an assembly relation, what does the # from the following symbol: (d#:#) mean?

The first # = dimension ID and the second # = Session ID

28. How can you put the dimensional tolerances into some relations?

By using the symbol "&" in front of G#, where # equals the geom tol ID.

29. What is an Evaluate feature and what is the difference between Relations / Evaluate and Datum / Evaluate?

Evaluate features can be created in both Part and Assembly mode and are used to perform measurements on model geometry. Evaluate feature is very useful to perform measurements between different components in an assembly. There are multiple measurements which can be performed by an evaluate feature varying from measuring the distance from a plane to a vertex to measuring the area of a surface.

When **Relations, Evaluate** is selected, Pro/ENGINEER prompts for a dimension symbol i.e. d4 d9, and returns the current value of that dimension. The **Feature, Create, Datum, Evaluate** function creates an "Evaluate feature" that will capture a measurement on the model and utilize that value in other relations.

30. In which case should you use relation in sketch (level) instead of relation in part (level)?

When you want to control the section with the help of trajpar sketch parameter (in the case of VarSecSweep), this one being evaluated by the system only at feature regeneration if the relation is at a sketch level. Also, it is necessary to use a relation at sketch level, when you want to obtain a relation with Know dimension.

31. How many times do you have to work in sketcher mode while you generate a Helical Sweep and why?

Twice. Once to sketch the trajectory and the second time to realize the



section.

32. What is the condition to modify the control graph for a Variable Helical Sweep?

The trajectory needs to be made of segments.

33. List all the Draft types that exist in Pro/E.

In order to create castings and dies it is necessary to use a draft form to obtain the inclination of the parts surfaces. The draft is a feature that can add or eliminate material. Because it is used with the intention of being able to extract the part from the die, it is recommended that it is built towards the end of the model, before realizing the parts' rays.

We must keep in mind that the draft feature is applicable only to cylindrical or plane surfaces.

Draft types:

Neutral plane

- no split: the system inclines the surface with a specified angle
- split at plane: the system "brakes" the surface at the intersection with the neutral plane, by applying the inclination angle to each of the sides
- split at sketch: the system "brakes" the surface along a contour that completely divides the surface and applies inclination angles to each side independently

Neutral curve

- no split: the system inclines the surface with a specified angle
- split at curve --> one side/both side --> independent/dependent; the system "brakes" the surface at the intersection with a neutral curve and the draft angles are applied to one or both sides, dependently or independently; this is a useful method when the curve corresponds to the die separation line
- split at surface--> one side/both side --> independent/dependent; the system "brakes" the surface at the intersection with another surface and it is necessary to have 2 neutral curves, one on each side of it. Draft angles can be applied on one or both sides, dependently or independently.

34. What is the difference between Simple Round and Advanced Round?

Advanced round is made of one or more sets of simple rounds. In most cases, when building the rounds on a model it is preferable to have "simple rounds" and only if the design requirements or simply the geometry of the part



requires so - to use the "advanced rounds".

In an "Advanced Round", each set of simple rounds can be attributed with round extent optionally, but the type of model attachment (solid or surface) can only be done globally for all sets.

Also, the "Advanced Round" allows you to define the feature: Round Shape -rolling ball/normal to spine; circular/conic- and to realize transitions between component sets, which determine the geometry that a round will have at its corners and / or ends.

Transition types for an advanced round:

Stop, Blend Srf, Intersect Srf, Continue, Corner sphere, Corner Sweep, Patch.

35. How can you determine the leader of a pattern?

The leader of a pattern is always the first one of the instances.

36. What are the advantages of using a Pattern Table?

Several design variants can be realized by adding more tables to the same patterned feature (transition from a variant to another is done with switch function).

A pattern table can be saved in a file and reused for another pattern table in a different model, insuring thus that the same configuration is maintained in different models.

37. What is the difference between Feature / Del Pattern and Group / Unpattern?

Feature / Del Pattern = allows you to delete a pattern and keep the leading feature

Group / Unpattern = allows you to delete a group pattern and keep the leading group

38. Can a current model use a Pattern Table that already exists?

Yes. Within option Modify Pattern Table, Pro/E allows for a pattern table initially used to be changed for an already existing one. Also, if there was a non-table pattern created within the model- the patterned feature can be redefined and the pattern changed into Pattern Table.

39. What are the advantages of using GROUP?

By Grouping you can obtain a set of features that will behave as a single entity. Several operations characteristic to the features can be performed on a local group: suppress, deletion, patterning. By group patterning some



configurations can be obtained that are impossible to realize otherwise.

If the local group was obtained by copying or patterning, a dependency is created between the original group and the copied or patterned one, which is kept also in the case of unpatterning or ungrouping. Using Make Independent (from Modify menu) can break the dependency.

40. How can you modify a single feature from a Local Group that has been patterned?

Unpattern and then Make Independent from Modify menu. A single dimension or the entire sketch can be made independent.

41. What is and how can you create a Family Table?

Family Table is an efficient method to rapidly and easily develop variants of the project, by creating part libraries and assemblies. In Family Table the variants are known as "instances", and the original model as "generic model". The use of this method leads to increased productivity through:

- keeping several similar models in a single file
- saving the various positions of a mechanism
- saving the various modeling phases of a model
- permanent saving of the model's variants by using Pro/Program.

The first step in creating a Family Table is to build the generic, which must contain all the possible features that can occur in the following instances. Also, it must contain all the valid relations for all instances.

After building the generic part, you must establish the dimensions, features and parameters that will be modified from an instance to another. It is possible to redefine the dimensioning sketches in order to reach the optimum variant.

Once we have established the elements to be modified, they must be introduced in Family Table with the help of the option Add Item. The system places each element in a column of the table.

After all elements are introduced in the table, you must use the Edit option from Family Tab menu in order to generate the instances. The system loads the table in the Pro/Table editor, the first row containing the values for the generic. In order to fill in the table, you can use several methods:

- manually
- by using the Patternize option from Family Tab menu
- by reading the data from a previously saved table

It is recommended in the end that you verify the instances created by using Verify from Family Tab menu.



42. What is and how can you create a UDF?

In order to save time and increase productivity, we can establish a library of common geometry by creating UDF-s. These are feature groups, references and dimensions saved for a future use in modeling. The features can be used to rapidly create models and also to enforce company standards.

In order to create a UDF, we must first build the geometry that will subsequently be saved. During its modeling, the parent-child relations that will be realized must be kept in mind. The features will be defined by using external common references or by referring one to another. The stages of realizing a UDF are the following:

- start to define the UDF and specify the option to keep it
- keep the reference part
- group naming
- form selection
- creation of prompts for external references
- definition of variable dimensions and elements
- finalizing definition

43. What are the differences between Local Group and UDF?

Local Group is a group created within a model which can't be transferred to another model.

UDFs are feature, references and dimensions groups, saved for a future use.

44. What is the difference between Reference option and Copy in Merge or Cut Out options (from Adv Utilities)?

By using the Merge- Reference function we can maintain the connection between parts. A Merge feature results in the part in which the operation is performed. In these circumstances, by modifying the main model, the one that contains the entire geometry, the system will propagate the modifications in each assembly component that was obtained with Merge.

In a Top-Down menu, you can use the Master Model technique in order to define the shape of one or more derived parts. This method is very useful in creating assemblies by defining multiple parts that have the same model at its basis, or parts created with the same geometry at the basis. The geometry is defined in a single part file and then the Merge/By Reference function is used in order to create the derived parts.

By using Merge - Copy function, the relation between the 2 parts is broken, and the system copies all of the part's features from the merged part.