Creo: Patterning and Mirroring

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Patterning: Allows the user to take a single feature and make multiple copies based on specific Dimensions. I.E Rivet/Rivet Holes on an airplane wing.

Types of Patterns

1. Dimension: Allows the user to select a locator dimension to determine direction of pattern
2. Directional: Allows the user to select an edge to determine direction of pattern
3. Axis: Allows the user to select an axis line to spin around to create the pattern
4. Point: Allows the user to select multiple points to copy the feature on
5. Fill: Allows the user to define a predetermined space using a sketch to fill within
6. Table: Allows the user to setup a spreadsheet table to determine x,y spacing
7. Curve: Allows the user to draw a curve to have the feature to follow

Pattern Points are calculated by the Centroid (Center of Mass) of what is being patterned. So, X,Y, and Z need to be considered when patterning.

Pattern Points Control: maybe skipped by clicking on the black dot that is desired to be skipped turning the dot White, which turns the patter off at that point

Mirroring: Allows the user to take a feature(s) and select a plane or surface and mirror (copy the object over
Design the following part

1. In the Model Tree Hold CTRL > Select Extrude 2 and Extrude 3 (or Hole1 Depending on how you made the hole) > Right Mouse Button > Group > Group

   NOTE: Grouping objects together allows the user to consolidate their model tree and keep features that are associated with each other together.

2. Rename Group: Select Local_Group > Right Mouse Button > Rename > Change it to Pattern

   NOTE: When renaming a group or any feature in Creo spaces are illegal characters and are not allowed. Name must be one string or use an underscore to separate out words
Patterning Dimension v Directional

1. Select Group_Pattern from the Model Tree > Model Tab > Select Pattern

**Select Type of Pattern**
- Direction 1 for Dimension or Directional Pattern
- Direction 2 for Dimension or Directional Pattern

**Pattern: Dimension**

1. Requires the user to select a dimension for the direction the pattern will go in

Select this dimension and the pattern will pattern in the direction of the arrowhead
2. **Direction 1:** Select the Dimension Tab

   - Number of Copies = 5
   - Includes original

3. **Direction 2:** Select the following
   - Click on Dimensions Tab > Select Direction 2 Box > Select Dimension
   - Increment = 1.25
   - Iterations = 3

**Dimension Tab**

**On-Center Spacing** between copied features

**On-Center Location.**

Point Represents the Centroid (Center of Mass) of the object. Notice how the points are not on surface but higher than the surface.
4. Green Check the Pattern. Finished Product
Pattern: Directional

1. Delete Pattern > Right Mouse Button on Pattern in the Model Tree > Select Delete Pattern (Be sure to select Delete Pattern and NOT Delete; Delete Pattern will save the original design)

   ![Select Delete Pattern]

2. Select Group Pattern > Pattern > Type Directional > Select the Edge Shown
   
   NOTE: Feature Group will disappear until an edge for direction is selected

   ![Select Edge]
3. Set the following
   a. Increment = 2.000
   b. Number of Iterations 3

4. Set Direction 2: Select the left Edge of the Part and Set the following
   a. Increment = 2.500
   b. Iterations = 2
**Pattern: Curve**

1. Delete the Pattern from Directional Pattern above
2. Select the Sketch Icon from the Model Tab
   a. Draw a curve using the spline tool (Spline is a free form curve be sure to keep the spline on the top surface of the part
   b. Rename Sketch 1 as Path
   c. Reorder the model Tree so the Sketch for the Spline is above Group_Pattern. To do this select the Sketch > Hold Left Mouse Button Down and Drag the Sketch above Group_Pattern
   d. Select Group_Pattern > Select Pattern Tool in Model Tab > Change Type of Pattern to Curve > Select the Path Sketch (Either on the part or in the model tree) > Set the following settings
   e. Green Check the Pattern
Pattern: Fill

1. Delete Pattern Curve
2. Select the Sketch Tool > Select Top Surface of Part > Draw a polygon that does not incorporate the total top surface > Green Check once done sketching
   See below for example
3. Rename the Sketch to Fill > Move the Fill Sketch above the Group_Pattern
4. Select Group Pattern > Select Pattern Tool in Model Tab > Change Pattern Type to Fill > Select Fill Sketch
5. Green Check
Finished Part

Pattern: Point

Point Setup should be setup based upon the Centroid of the object being patterned (Center of Mass or Volume) A datum May need to be setup at the center of the patterned feature.

1. Setup Datum at center of feature Click on Model Tab > Select Plane > Select Top Surface > Translation .500 > Select OK

2. Select Model Tab > Select Datum Points > Select Datum Created in Step 1 Pattern: Point
3. Select the Datum To place Points > Use the Green Hand Holds to Locate Points (Similar to the Hole Tool)

4. Model Tree: Move Datum and Point Features above the Group_Pattern

5. Select Group_Pattern from Model Tree > Select Pattern Icon in the Model Tab > Change Type of Pattern to Point > Select Datum Point Option > Select Datum Point Feature from Model Tree > Click Green Check
Completed Pattern

Points Not Setup at the Centroid of the Original Object
In this example the points are initially setup on the Top Surface of the rectangular Box. Notice the height difference between the original on the left side and the three new iterations. This happened because the point setup was on the top surface of the block making that point the centroid of the new iteration in the pattern.
Pattern: Axis

Axis Pattern allows the user to create a pattern based on an axis line > Creating a pattern that follows a curve.

Part

1. Turn Datums On > Select the Hole Diameter .375 > Select Pattern > Change Pattern Type to Axis > Select Center Axis of Part

Mirror

Mirroring allows the user to select multiple features from an object and translate it over a surface or datum creating an exact copy of the object. For Example

Select a Surface

1. Select all Extrude 1, Group_Pattern, DTM1, and Datum Point Feature from Model Tree > Mode Tab Select Mirror > Select the Front Face of the Object > Green Check
Select a Datum

1. Create a Datum offset from one of the faces.
2. Select Features to be mirrored from Model Tree > Model Tab: Select Mirror Icon > Select Newly Created Datum > Click Green Check

Notice the Offset Distance is the same for the new copy on the right side of the picture.