Pattern
Dimension Pattern
Identical Pattern: Bidirectional Linear Pattern
Box and A Hole

1. First, create the base block by Sweep the open section along a rectangular trajectory as shown below.
2. Create a hole by Extrude at a position shown below and make dimensions as shown relative to bottom and right planes. These dimensions will be used to specify the increments in patterning (if wrong, it will fail).
Linear Pattern

- Click the feature (i.e. hole) to pattern. This action activates the pattern menu.
- Click pattern menu to get into pattern > Now you see the pattern dimensions and pattern user interface that consists of the following items: Feature icon, Dialog bar, Slide-up panels, Shortcut menus.
On entering pattern, the default direction interface is active for first direction > pick on the horizontal dimension > enter x-increment and may change the item number.

Click the second direction interface to activate > pick on the vertical dimension > enter y-increment and may change the item number.
Now, if you want to see the increment dimensions and values, click the dimension slide-up window. You should see as left. You can close it by clicking once again.

You may modify values or delete the selection by highlighting and using [Delete] keyboard.

Select the pattern option by [Options > Identical]

Done
Dimensions

[Diagram showing dimensions of a square object with annotations: 40.00, 30.00, 2 H O L E S, Ø 10.00, 2 H O L E S, and 30.00.]
Problem 1

Problem. Create L-shaped bracket by **Extrude** and a rib by **Rib**. Then, use **Pattern** to make the part. The first rib at the left is at distance 1.0 from the end with thickness 0.4.
Problem 2

Problem. Add the increment of the hole in both directions by 5 and 10 to create the following pattern. What is the size of the largest? Use [Analysis > Measure > Diameter > pick the hole].
Radial Pattern

As the linear pattern used linear dimensions for pattern increments, the radial pattern needs radial dimension. The following shows the steps to create the feature.
Revolve

1. Create the wheel by Revolve. The dimensions of the section are given above.
Hole

2. Create a hole by Hole. Hole > Simple / Diameter 30 / Thru All
3. Select the plane where the hole to be placed as primary reference
4. Click Placement tab > Radial (for the type) > click Secondary references (to activate) > Placement Type = Radial (click the menu place holder to see all menus as shown below) > pick on the axis at the center > ctrl + pick on the Front DP for another secondary reference > change the radius 143 and the angle to zero (see below) > Done (feature).
Select up to 2 references, such as plane, surface, edge or axis to define hole offset.

Primary:
- Surf.F5(PROTF)
- Flip
- Radial

Secondary references:
- A_2(Axis):F...
- FRONT:F3(D...)

- Radius: 143.00
- Angle: 0.00

FRONT
PRT_CSYS_DEF
Protrusion id 60
Insert Here

KIM, ME, NIU
5. Radial Patterning. Pick on the hole (to highlight) > (Edit >) Pattern
6. Pick on the zero angle dimension for hole > enter 45 for angle increment > CR > enter 8 for first item number.
7. The default pattern type is General in the Options interface.
8. If needed, the pattern can be also done in the radial direction by [click the second item interface (to activate) > pick on radial dimension > enter -50 > CR > enter 2 for second item number]. This will not be visible buried in the feature, though.
Select dimensions to vary in the second direction.

Direction 1

<table>
<thead>
<tr>
<th>Dimension</th>
<th>Increment</th>
</tr>
</thead>
<tbody>
<tr>
<td>d22:F7(HOLE)</td>
<td>45.00</td>
</tr>
</tbody>
</table>

Direction 2

<table>
<thead>
<tr>
<th>Dimension</th>
<th>Increment</th>
</tr>
</thead>
<tbody>
<tr>
<td>d21:F7(HOLE)</td>
<td>-50.00</td>
</tr>
</tbody>
</table>
Mirror

9. Mirror. Edit > Feature Operations > Copy > Mirror / Select / Independent > Done > pick on the model > Done > pick on the back surface of the model as the mirror plane > Done
Problem 3 (Lug)

Create the lug in the figure following the procedure below. Here, the radial pattern dimension cannot be created directly from the existing geometries. Thus, the intermediate feature, datum plane, is used to create the necessary radial dimension for pattern.
1. First create the base table.
2. Then, create a rectangular bar and a cylinder on a datum plane at the distance of one off the top surface with depth by ToNext toward the upper surface of the table.
3. Create patterns as in the previous sections.
Dimensions
Unidirectional Linear Pattern

1. First create the base table and a hole. Create the block by [Extrude] with dimensions of 20 x 15 x 3 and a hole by [Hole] with diameter of 1 at 3 and 2 distances from left and bottom, respectively.

2. Pick on the hole (to highlight) > Pattern > pick on the horizontal dimension > enter the increment 3.5 > CR > ctrl + pick the vertical dimension > enter 2.5 > CR > enter 3 for the number of items > Options > Identical > Done (feature).

4. Select the pattern in the model tree > use MRB and select Edit Definitions in the popup menu > click Dimensions interface > ctrl + pick on the diameter dimension > enter the increment 1.5 > CR > click Options interface > select Varying > Done (feature).

5. Once again, use Edit Redefinitions of the pattern > change the number of items to 4 > Done (feature).

6. What happened? The answer is the failure of the pattern creation because the holes would overlap. Only the General pattern allows overlapping patterned features. Most failures of the pattern generation are due to this overlapping.

7. Undo Changes (or Fix Model > Feature > Redefine > Select > select the pattern in the Model Tree > change Options to General > Done (feature) > Yes. Now, the pattern has overlapping holes.
Pattern with Relations

1. First create the base table.
2. Then, create a hole at lower left corner with appropriate dimensions for patterning.
3. Pick on the hole > Pattern
4. Click **Dimensions** interface > pick on the horizontal dimension > check **Relation** check box below the dimension box > click **Edit** > enter the equation as shown in the first figure below > save and exit the file
5. Ctrl + select the second dimension > pick on the vertical dimension > select **Relation** for option > Edit > enter the relation as in the second figure below > save and exit the file > enter number of items > **Options** > Identical > Done
   Alternatively, click the second dimension pane to activate > pick on the vertical dimension > click **Relation** check box > Edit > enter the relation as in the second figure below > save and exit the file > enter number of items > **Options** > Identical > Done
5. Edit Definition > Options > **General** > ctrl + pick on the hole diameter > enter the increment of one > CR > Done (feature). Now, the pattern appears as the second figure above.
Modifying Relations

1. Double click the hole other than the first one to change the dimensions.

2. If the relations are to be edited, click the pattern in the model tree and on top of the highlighted pattern, click MRB and hold down and choose Edit Definition > click Dimension interface > click the dimension with relation > Edit > modify the relations > save and exit > Done > Regenerate.
Group Pattern and Length Ratio
1. First, create the basic feature by Sweep (see the dimensions in the figure below). For section, sketch the lower edges first and then use [Offset] to offset them by 0.25.

2. [Divide] the trajectory at midpoint of the larger arc (for future use in datum curve). This division must be done now that will be later used in creation of datum curve for pattern.
Selecting a chained curve by Chain for the type

- Select the first line segment, (1), and the last segment, (2).
- Depending on the case, the tangent chain may select all between them or not.
- If the whole chain is selected, then the starting point of the first line becomes the starting point of the chain.
- If only two line segments are selected, the starting point is the starting point of the first segment in the chain direction (1) to (2) as above.
- If “Next” or “Previous” is used to select the whole chain in the previous case, the direction reverses from the last point (now, this is the starting point) if two line segments are different kind (i.e., line and arc). If two segments are same kind like arc and arc, then the second segment becomes the first and its starting point becomes starting point of the chain.
- Accept > -1 > Done (sketch) > Done (feature).
3. Create the datum curve for pattern along which the pattern will be created by
   • Sketch tool > select the flat top surface as sketch plane and one sketch reference plane > pick on the edge views of two datum planes as references > Sketch
   • Offset Project > click Chain for the type > select the first line segment, (1), and the last segment, (2).
   • Depending on the case, the tangent chain may select all between them. If this is the case, then the end point of the first line becomes the starting point of the chain. If only two line segments are selected, the starting point is the starting point of the first segment in the chain direction (1) to (2). If “Next” or “Previous” is used to select the whole chain in this case, the direction reverses from the last point (now, this is the starting point).
   • Accept > -1 > Done (sketch) > Done (feature).
4. Before placing a hole at mid-section, place a datum point at the position [Make sure to have selected the curve just created > Datum Pt > pick on the curve > see the length ratio value of zero > OK]

**Hint:** The direction of a chained curve can be switched by Next End in the dialog above.
5. Now, place a datum axis at the point by [click the datum point just created if not highlighted > Datum Axis > (the datum point already appears in the dialog) ctrl + pick on the top surface > Ok (see above)], and then use the Hole menu to create a hole by [pick on the axis at the point > enter the diameter of one > change the depth to Thru Next Surface > click Placement interface and see the hole type Coaxial > click No Items in Secondary references to activate > pick on the top surface > Done (see below)].
6. Create a group [Edit > Feature Operations > Group > Local Group > enter a name > pick on the datum point, datum axis, and the hole on the model tree > Done > Done > Done > Done].

7. Create a group pattern [click the group in the model tree if not yet highlighted > Pattern > pick on length ratio dimension (i.e., 0.00 REL) > enter the increment (i.e., 0.1) > enter total number in pattern (i.e., 10) > Done. Note the length ratio ranges from 0 to 1 in CCW direction.

8. Now, hide the datum curve to finish the model by [click the curve in the model tree > click and hold down MRB > Hide].

**Note** that only the datum point menu has the Length Ratio menu. If you want to utilize the length ratio option, you must use datum point.
Placing a Point on a Curve

1. **Sketched curve** must be preselected if that’s the curve to place a point. It cannot be selected in Datum Point dialog.
2. When clicking a curve to place a datum point, if user click the end point, then Offset value is grayed and the Ratio value of the point on the curve is **unavailable** for future like dimension pattern. Click a point in the middle of the curve and enter the value zero to make ti available for future.

![Datum Point dialog]

**References**
- Curve: F7(SKETCH_2) On

**Offset**
- 0.00

**Ratio**

**Offset reference**
- End of curve: Next End
- Reference: Click here to a...
Problem 4

Create a path and place a cylinder at a position on the path and then pattern with Curve pattern.
Problem 5

Create the following bevel gear with appropriate dimensions.
More Types of Pattern

First, create a plate and then use Editing > Offset / with Draft > to draft two rectangles as below.

Note that two drafts above can be also done by Offset / Expand. However, this allows only two types of pattern, Dimension and Table.
Pattern / Direction
Pattern / Axis

This pattern allows the pattern even in radial direction.
Pattern / Fill

Define internal sketch to fill by References > Sketch
Pattern / Curve

The previous (dimension) group pattern can be done by Curve pattern. This may be easier in some cases.
Pattern / Point

Click Datum Point option as below and create them.
Pattern / Table

A pattern table can be created to specify the patterns for more complex patterns.
Any dimensions can be chosen to vary. Below only two dimensions are selected. Save and exit the table.
There are three Offsets with two added by pattern.