Rotational Patterns of Sketched Features
Using Datum Planes On-The-Fly

Patterning a sketched feature (such as a slot, rib, square, etc.,) requires a slightly different technique. Why can’t we create a sketched feature just like we did the Insert – Hole and pattern it? First, we are creating a rotational pattern. That means we need a distance from an axis as a diameter, and the distance between the patterns in degrees. When we created the hole using the Simple/Sketched – Diameter options, we entered X° from the edge view of a principal plane that intersected the cylinder’s axis, allowing us to use that dimension (angle) to rotationally pattern the hole about the cylinder’s axis. What other options do we have when we need to sketch a shape, rather than use Insert – Hole?

- You could try sketching the shape as usual, but you aren’t given an opportunity to locate the shape from the edge view of a principal plane using degrees – so this method won’t work.
- You could try sketching the shape as usual, but add a centerline through the axis of the cylinder and the shape. While this method appears to work, it is unpredictable. Very seldom is the correct number of patterns displayed. Note: "Do not use the angular dimensions that controls centerlines of sketched features for rotational patterns. The results are not predictable." Part Modeling Users Guide, Release 20, p14-9.
- You could try creating an angled plane before creating the pattern leader, but when you try to pattern the pattern leader, you can’t see the angle in order to select it.
- Therefore, you need to create an internal datum plane, called on-the-fly, so you can select its angle for rotation.
- **No matter what** - do **not** start with the datum plane created on-the-fly at 0°.

Two examples on sketching a shape, a slot and a rib, will be discussed. The slot will be constrained to the edge view of the datum created on-the-fly and the rib will be sketched directly on the datum created on-the-fly. Both would be patterned the same, so only the slot will be patterned to create a total of 6 slots equally spaced.

Create a simple ring with a diameter of 10, thickness of 1, as shown by making an extruded protrusion.

Can you constrain the pattern leader (slot) to the edge view of the Profile plane and then pattern it? First ask, when I display dimensions for the Profile plane will there be an angled dimension for me to use? The answer is no, therefore a datum-on-the-fly must be inserted. Select Extrude icon – select appropriate extrusion distance, i.e., Thru All – if a cut select Remove Material icon – Since we don’t have a datum plane that will display an angle dimension we need to create one on-the-fly. Select Datum Plane Tool – select axis A_1, Through – press the CTL button and select plane to angle from, i.e., Profile, Offset Rotation – type in degrees to rotate, i.e., Enter value, key in the angle, i.e., 30 (you can use any angle but 0.) – Ok – Resume icon – Section icon – select the sketch plane (see examples below) –
Select **Sketch button** –

If you made the correct selections, the DTM1 plane will be in a vertical position.

References: Select two planes perpendicular to the plane you are sketching on, or a plane and an axis, depending if you are sketching perpendicular or parallel to the inserted plane. (If your datum plane revolved clockwise instead of counterclockwise, don’t worry – it can be fixed later.) – **Close button**

**SLOT**

References:
- DTM1
- Horizontal plane

**RIB**

References:
- Front of the cylinder
- Axis to revolve about

**Sketch shape.**

**Slot.** Constrain the center of one of the arcs to the edge view of the datum created on-the-fly. Remember to dimension following ANSI standards. The bolt circle dimension is from axis $A_1$ to the center of the slot (right mouse click and use **Pick From List** to assure this) and given as a diameter.

Note: this figure does not follow ANSI dimensioning standards and DTM1 is not parallel to the right side of the screen.
Rib. Sketch on the datum plane created on-the-fly. Remember to dimension following ANSI standards.

Extrude the result should be as shown:

Pattern this as the activity before, selecting on the angular dimension for the First patterning dimension. 60 degrees, 6 instances. (as instructions requested.)

Edit the part file to move the pattern leader to TDC (top dead center). (Can't remember which angle to pick, refer to the last page.)
Edit the part file to 8 equally spaced slots, as shown below. (Can't remember which angle to pick, refer to the next page.)
Right mouse click on the Pattern Leader and select Edit to see the dimensions for the primary slot.

If you can’t remember which slot is the Pattern Leader, select the first slot listed in the Model tree.

Use this dimension to orientate the Pattern Leader.

Internal datum plane – used to orientate the Pattern Leader from the Profile Plane.

Internal datum plane – used to orientate the Pattern Leader from the Profile Plane. Set to 0° in previous step.

Use this dimension to set the number of patterns, including the Pattern Leader.

Use this dimension to set the distance, in degrees, from the Pattern Leader.