The Curve Driven Pattern places instances along a curve such as a sketch, a model edge, or a composite curve. A Curve Method determines how the curve is used. An Alignment Method specifies how the seed feature is mapped to the curve.

1. **Open the part Curve Driven Pattern.**
The countersunk hole is the feature that will be patterned.

2. **Sketch.**
Rollback before the M4.5 Diameter Hole feature. Open a sketch on the front face of the cam and create a 5.5mm offset of the outer profile.

3. **Exit the sketch.**
Roll forward to the end of the feature tree, rebuilding the part.

   Rename the sketch Curve Sketch. This is the curve that will drive the pattern.

4. **Locate the hole.**
Expand the M4.5 Diameter Hole feature. Edit the sketch that contains the locating point. Add two geometric relations as follows:

   - **Horizontal** with respect to the origin.
   - **Coincident** to the Curve Sketch.
5 Curve driven pattern.
For Direction 1, select Curve Sketch from the FeatureManager design tree.

For the Features to Pattern, select the M4.5 Diameter Hole feature.

Set the Number of Instances to 20 and select the Equal spacing check box.

6 Click OK.
Hide Curve Sketch, the sketch that was used to drive the pattern.

The completed pattern is shown at the right.

7 Save and close the part.