- Revolved feature require axisymmetric geometry and a centerline (used as the axis) in the sketch. Revolved feature can be a base, boss or a cut. The axis can be a centerline, line, linear edge, axis or temporary axis.
- Geometry for revolved feature is created using the same tools and methods as extruded feature.
- Special rules apply to sketches for revolved features:
  - A centerline or sketch line must be specified as the axis of revolution. This axis must be selected before creating the revolved feature,
  - The sketch must not cross the axis.

**LAB EXERCISE:**

1. Create a disc by revolving its profile about an axis.
2. Use the dimension provided. **Units: inches**
3. Use relations wisely to maintain the design intent.
4. Optional: Text in a sketch. See attached handout

![Diagram of a disc with dimensions](image)

**Procedures:**

- Create a new part using inches as the unit (use Inch template if available).
- Select the front plane as the sketching plane.
- Select a Revolved boss/base feature icon (or Insert > Feature> revolved boss/base; from toolbar menu).
- Draw a vertical centerline through the origin point.
- Starting a short distance to the right of the centerline, draw a cross section of the disc profile using the given dimension. Do not yet include the fillets; they will be created as separate features after revolving.

**Note:** The distance from the centerline to the first line on the cross section determine the size of the center hole.
- When the sketch is completely defined, meaning lines and curves are black, click the sketch button off.
- Fully revolve the profile, 360 degrees, using the centerline as the axis of revolution.
- Add fillets as required to disc.
- Create a hole on the disc and use circular pattern to make 7 copies.
Optional: Text in a Sketch

Text can be added to a sketch and extruded to form a cut or a boss. The text can be positioned freely, located using dimensions or geometric relations, or made to follow sketch geometry or model edges.

Introducing: Text Tool

The text tool allows you to insert text into a sketch and use it to create an extruded boss or cut feature. Since SolidWorks software is a true Windows application, it supports whatever fonts you have installed on your system.

Where to Find It

- Click **Tools, Sketch Entities, Text**.
- Or, on the Sketch toolbar click **Text**.

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1. **Construction geometry.**
   Sketch on the front face and add construction lines and arcs as shown.

   Tip
   Use *Symmetric* relationships between the endpoints of the arcs and the vertical centerline.

2. **Text on a curve.**
   Create two pieces of text, one attached to each arc. They have the following properties:
   - **Text**: Designed using
   - **Font**: Courier New 11pt
   - **Alignment**: Center Align
   - **Width Factor**: 100%
   - **Spacing**: 100%
   - **Text**: SolidWorks
   - **Font**: Arial Black 20pt.
   - **Alignment**: Full Justify
   - **Width Factor**: 100%
   - **Spacing**: not applicable when using Full Justify

3. **Extrude.**
   Extrude a boss with a **Depth** of 1mm and **Draft** of 1°.

4. **Save the part and close it.**