Creating New Dimensions

Insert > Dimension

Use **New References** when creating single dimensions

Use **Common References** when creating baseline dimensions

Choose the option for attaching the dimension to the draft entity. A different option may be used for each extension line attachment point.

- **Entity:** Will allow the user to attach the dimension to the endpoint of lines, arcs and to the tangent point of curves. To use, pick the line or arc nearest the endpoint desired.
- **Midpoint:** Will allow the user to attach the dimension to the midpoint of lines and arcs. To use, pick the entity.
- **Center:** Will allow the user to attach the dimension to the center of arcs and circles. To use, pick the arc or circle.
- **Intersect:** Will allow the user to attach the dimension to the intersection point of lines, arcs and circles. To use, pick two entities.
- **Make Line:** Will allow the user to create a line to which the dimension will attach.

When an attachment point is selected, the point will be marked with a small crosshair symbol.

Move the pointer to the desired location for the dimension and press the MMB

A menu will open prompting the user to define the dimension orientation (Horizontal, Vertical, etc.)
* Alternative for the **Entity** mode. If the option is select the user may also create Linear Point-to-Point dimensions, Radial dimensions and Diametric dimensions using the same techniques as within the Sketcher.

  To create the point-to-point dimension, pick a line and then place with the MMB.

  To create a radial dimension, pick an arc then place with the MMB.

  To create a diametric dimension, double-pick on a circle or arc then place with the MMB.