

## Dimensioning Improvements

SOLIDWORKS 2016 improves the behavior of selections when you add a dimension to a sketch

If you select an endpoint or sketch point and then accidentally click in the graphics area, the *first* selected point is retained so you can try again to select a second entity.

If you select an invalid entity to complete a dimension, such as a spline, ellipse or parabola, instead of an error dialog box, a tooltip warns that a dimension cannot be created from the currently selected items.

Move the pointer to dismiss the message. Your first selection is retained so you can select a second entity.

