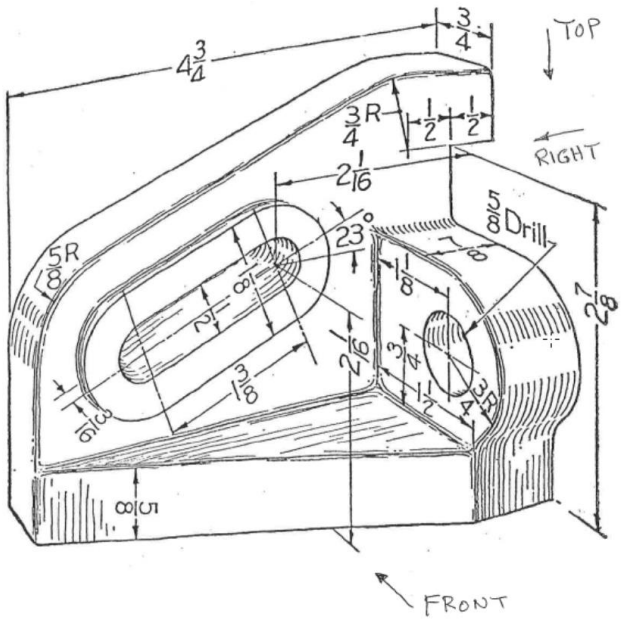
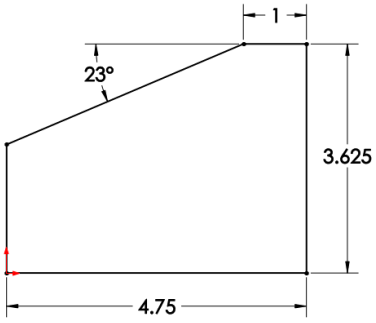
Recall that the design engineer provided the following dimensioned isometric sketch

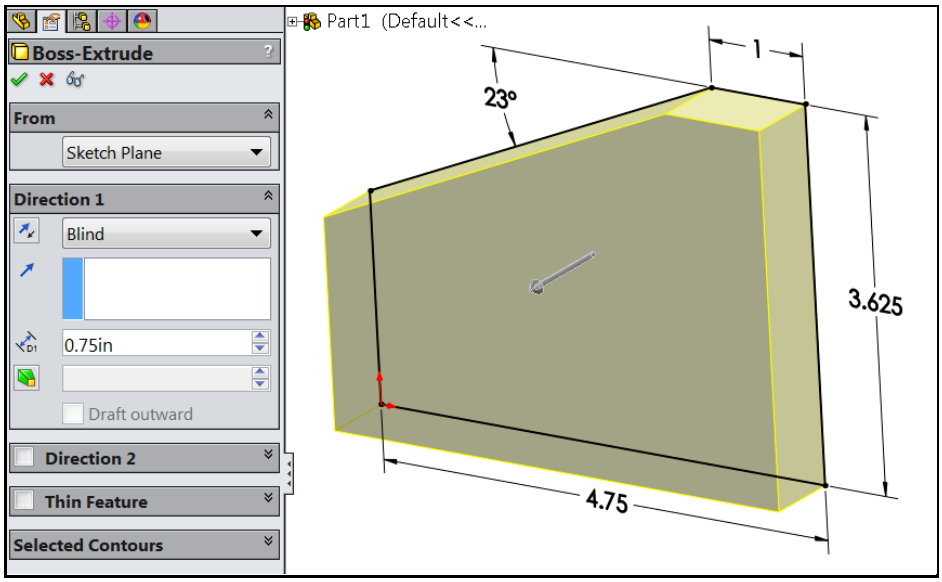


The above "design intent dimensions" are the only ones that should appear in the final mechanical drawings. Therefore, they are the dimensions to be used in all part feature sketches.

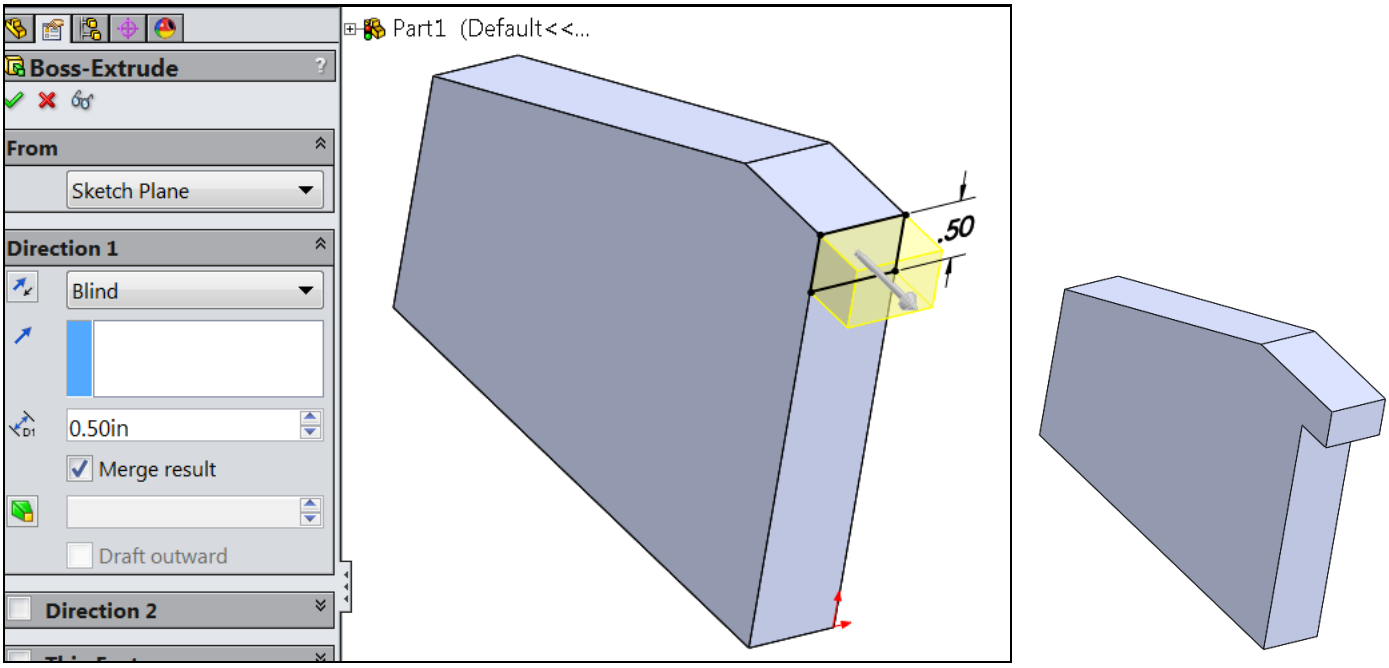
One approach to building the sketch part:

(Right click on the Front plane name and select insert sketch.) Sketch the back feature on front plane → Features → Extrude

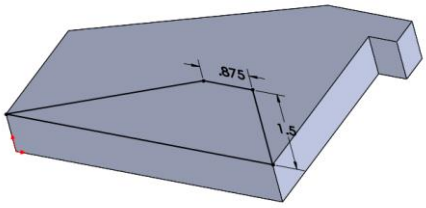


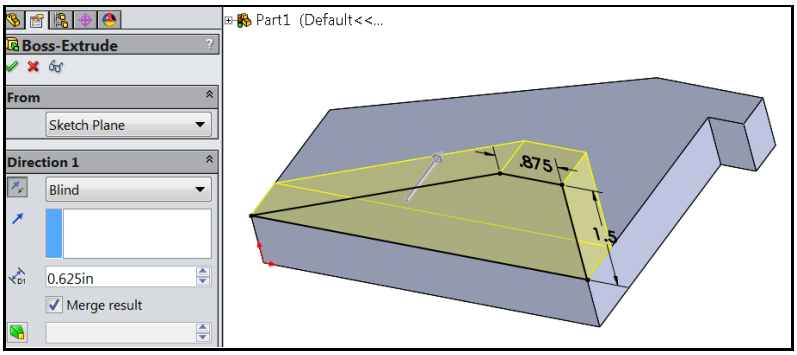


Attach the overhang by sketching on end surface: (Right click on the side surface, select Insert Sketch.) Rectangle → Smart dimension → Feature → Extrude

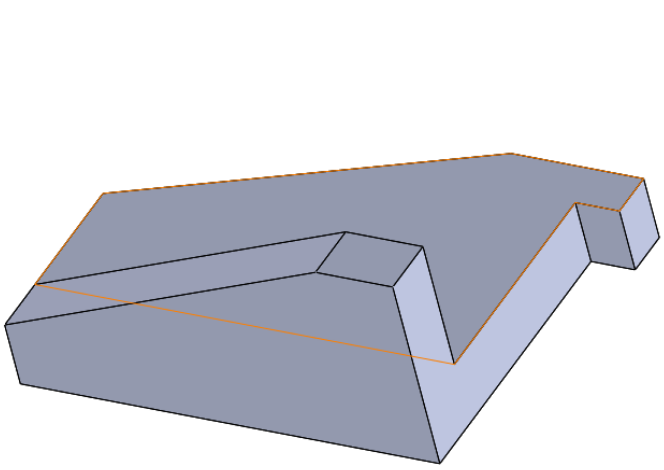


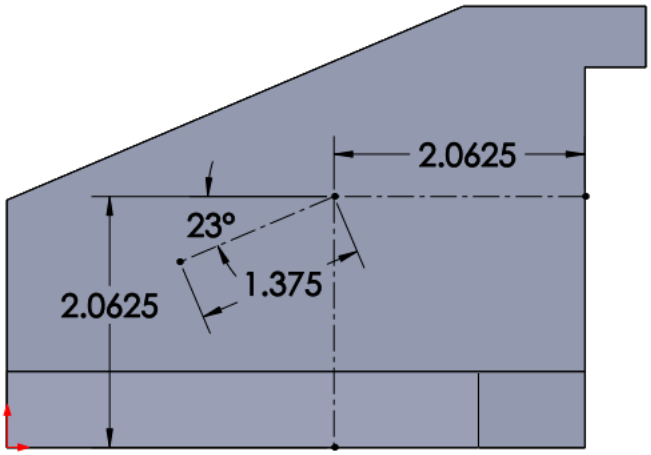
Sketch on bottom surface Lines (form trapezoid) → Smart dimension → Feature → Extrude



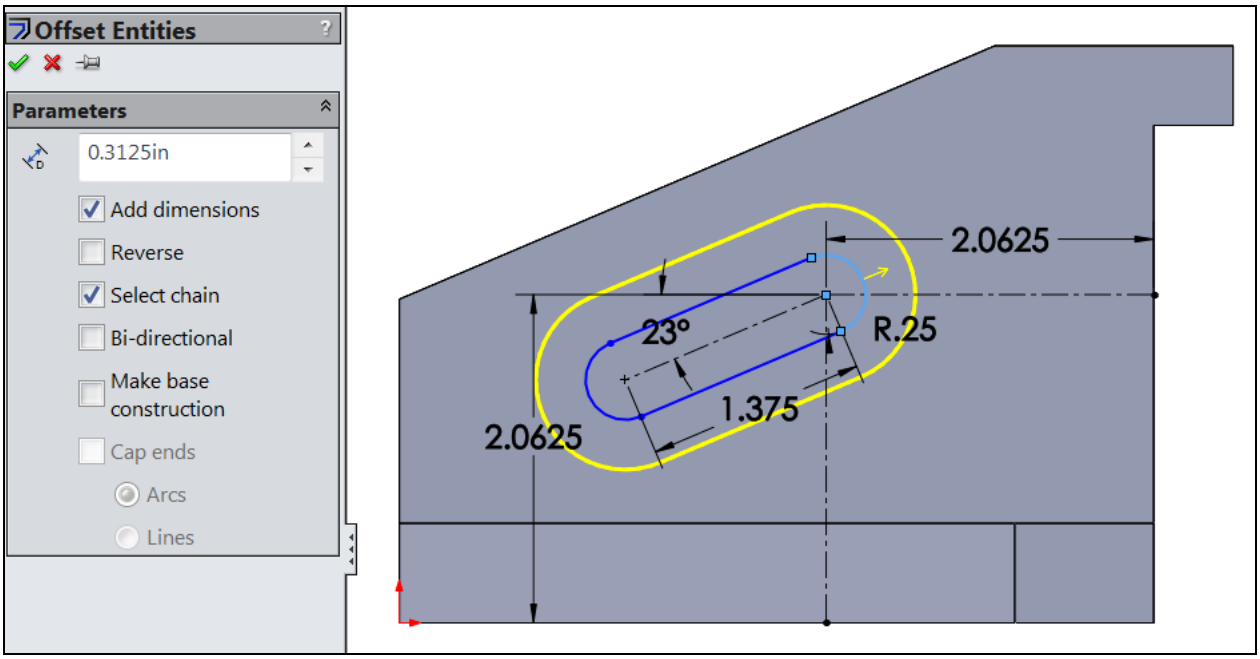


Sketch on front surface and add construction lines for racetrack features

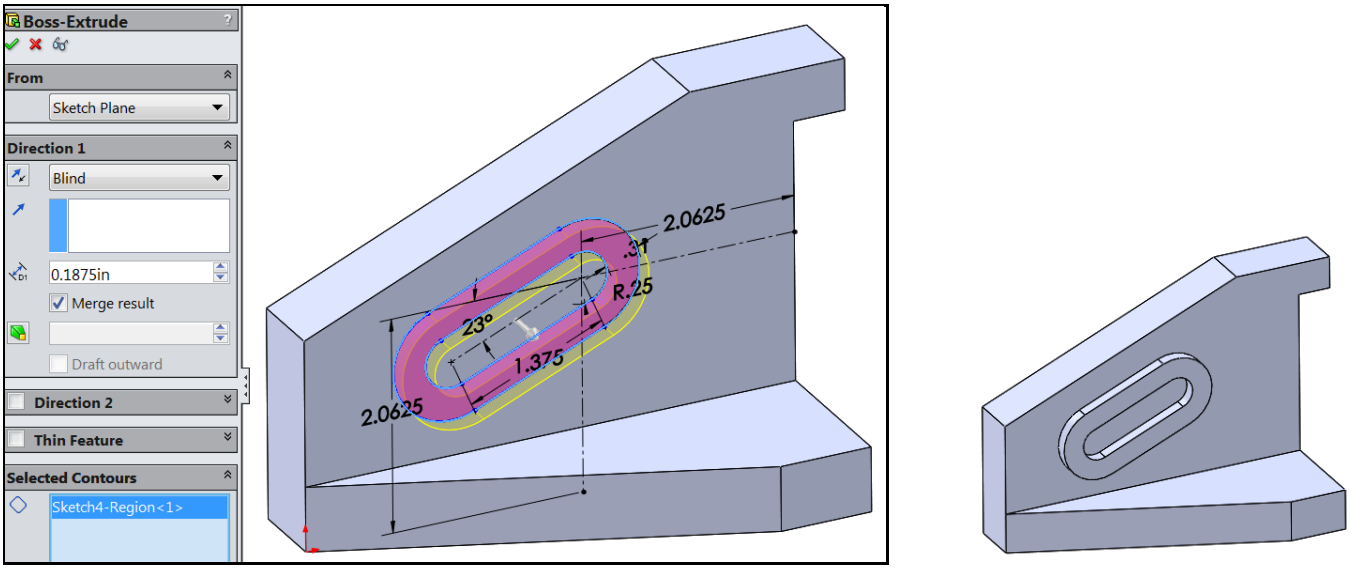




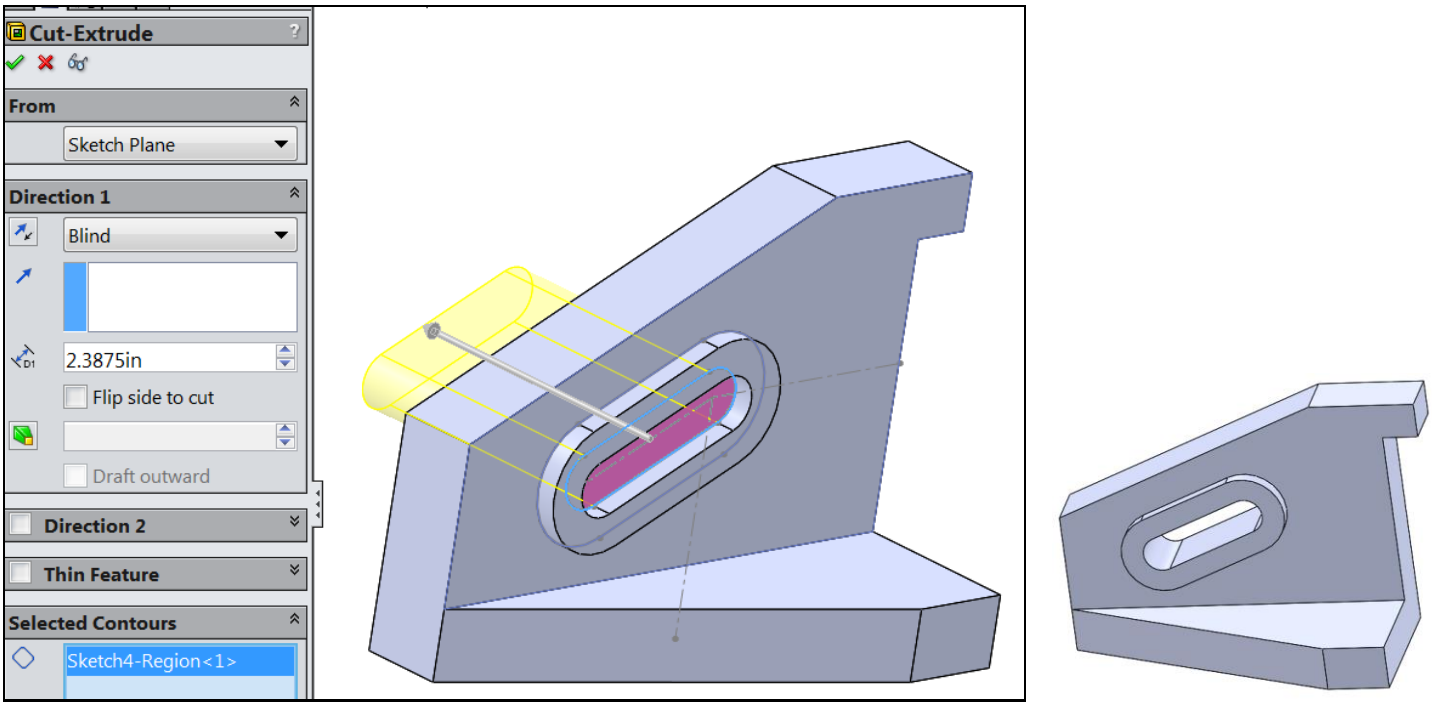
On front surface insert inner two arcs and two lines Offset Entities (needed amount) → Feature → Selected Contours



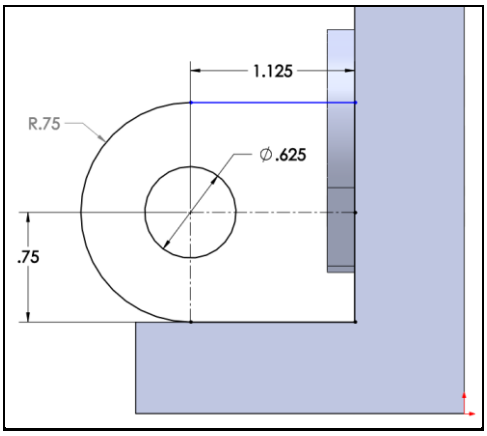
Select outer area to be extruded, set amount

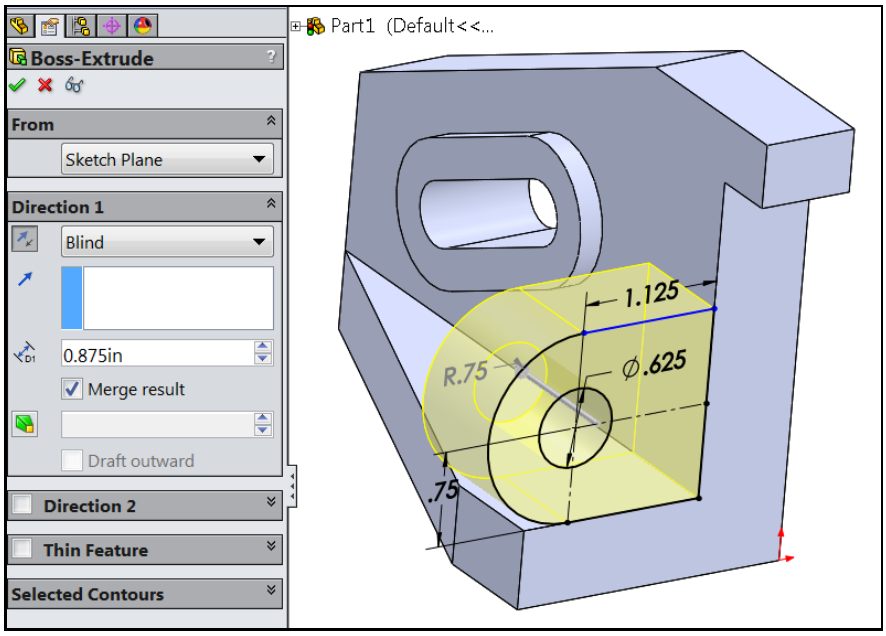


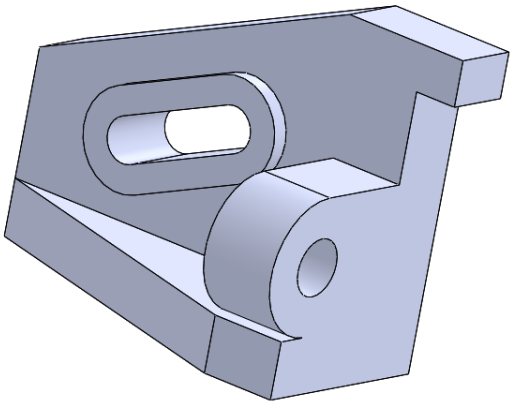
Select the same sketch again: Feature → Extrude Cut → Selected Countours → pick the inner oval



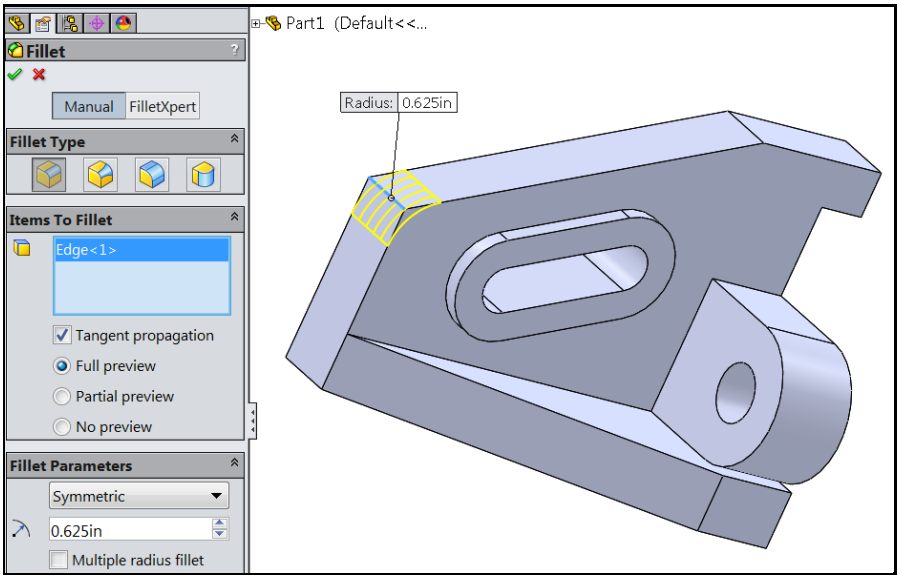
Build the curved end with the circular hole: Right click on the side surface and insert sketch. Use lines, circle and an arc to form the closed area and the extrude it. (Note the sharp re-entrant corner to be fixed later.)

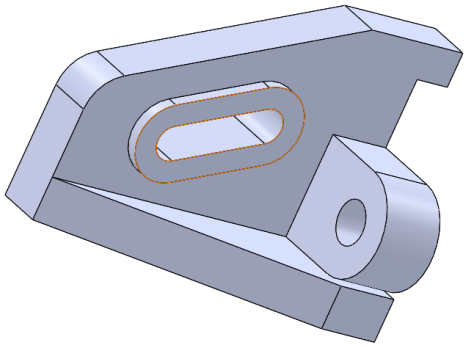




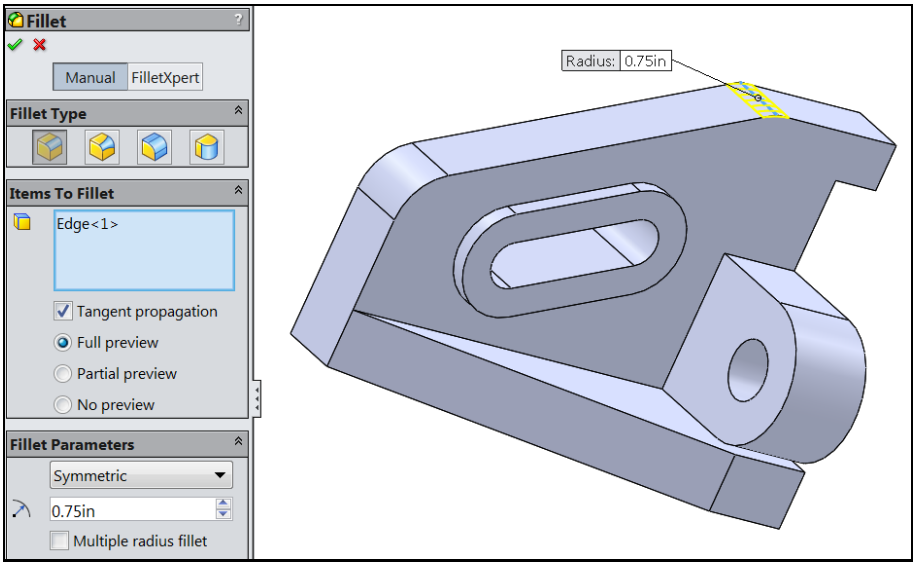


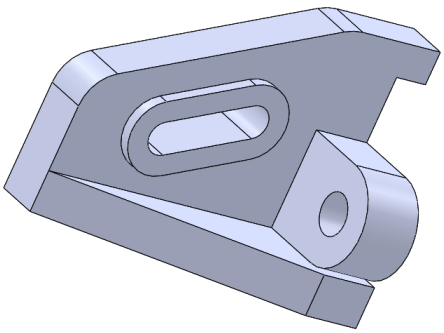
Now put in solid fillets (rather than including them in the two-dimensional sketches). Fillet → pick the line → set the radius

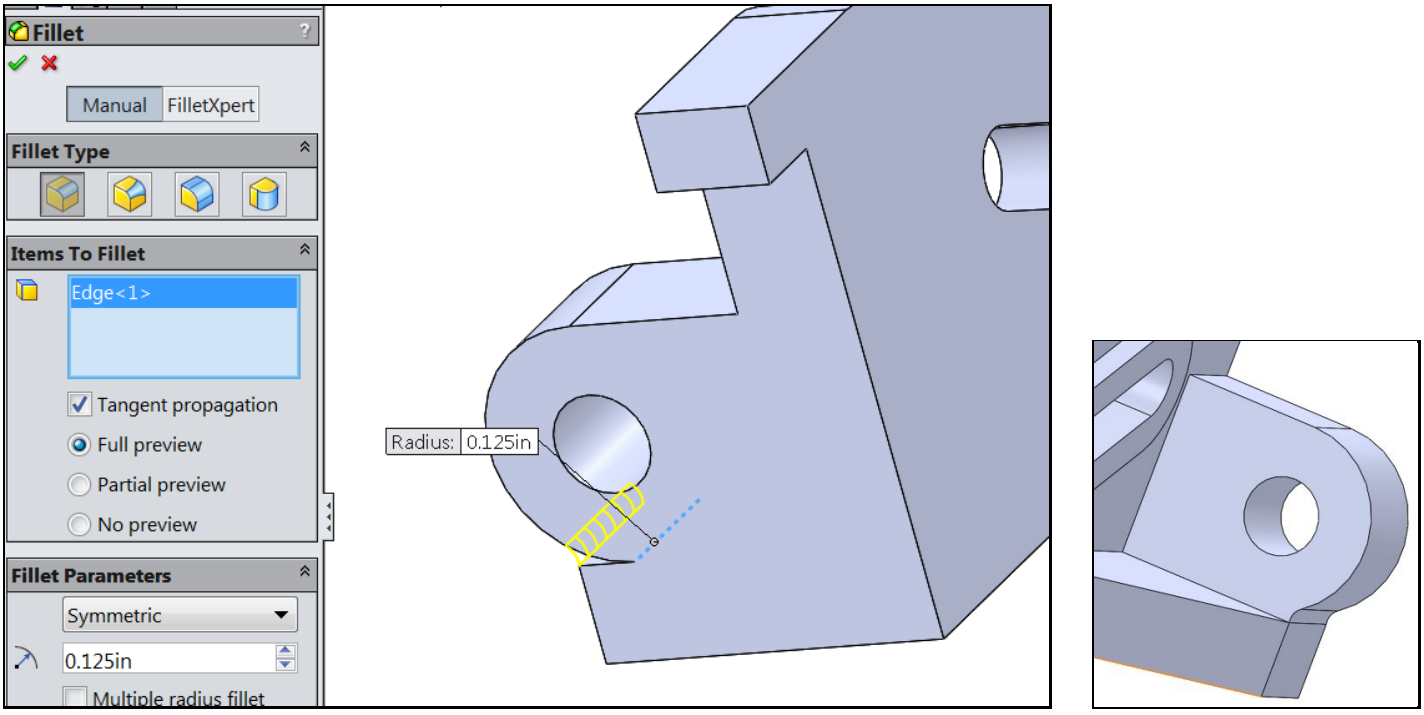




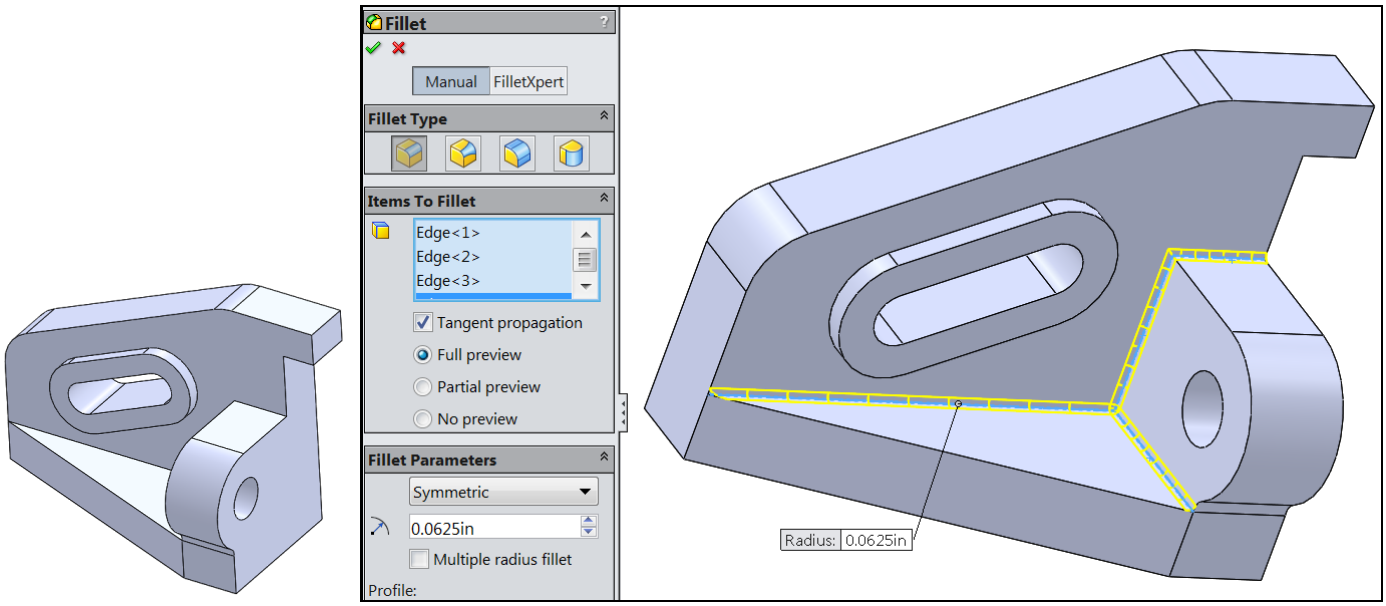
Repeat as needed for different radii:

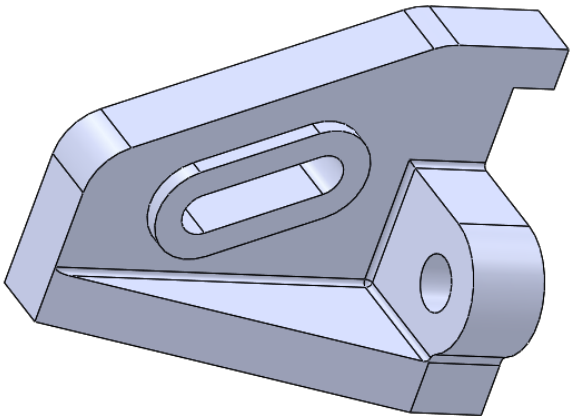






Fillets also can apply to multiple lines





A slow double click allows you to rename each feature after you have built it. This makes it easier to go back and make changes or corrections.