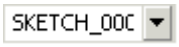
Explanation of buttons used for sketching in Unigraphics

Sketcher Tool Bar

Finish Sketch is for exiting the Sketcher Task Environment.

Sketch Name is the name of the current active sketch. You can also use this to change the name of a sketch.



Reattach is for attaching an existing sketch to a different face or plane.

Orient View to Sketch is for orienting the view to be looking directly at the sketch plane.

Orient View to Model is for returning the orientation of the view to the one displayed prior to entering the Sketcher Task Environment.

Create Positioning Dimension is for creating, editing, or deleting positioning dimensions.

Update Model will update the model, using the changes made to the sketch, without leaving the Sketcher Task Environment.

Delay Evaluation is for delaying the evaluation of the active sketch. Normally a sketch is evaluated as you work on it.

Evaluate Sketch is for asking the system to evaluate the sketch. This is only available when Delay Evaluation is on.

Sketcher Operations Tool Bar

Mirror is for creating a mirror image of selected sketch curves, about a line.

Offset Extracted Curves is for creating a set of curves that are offset from curves extracted into the sketch.

Edit Curve is for editing sketch curves.

Edit Defining String is for changing which curves in a sketch are used to generate a specific feature.

Add Objects to Sketch is for adding non-sketch curves to a sketch.

Add Extracted Curves to Sketch is for creating an associative copy (within a sketch) of curves/edges that are outside the sketch.

Sketch Constraints Tool Bar

Dimensions is for creating and editing dimensional constraints.

Create Constraints is for creating geometric constraints.

Automatic Constraint Creation is for creating many constraints at once.

Show All Constraints will display all created geometric constraints on the graphics screen.

Show No Constraints will remove the display of all created geometric constraints.

Show/Remove Constraints is for listing and/or removing geometric constraints.

Animate Dimensions will vary a selected dimension graphically.

Convert To/From Reference is for changing geometry or dimensions to or from reference. Reference dimensions are not used to evaluate the sketch. Reference geometry is not used for creation of features.

Alternate Solution will find the other solution of a dimension, or the other solution between an arc/circle and a line.

Infer Constraint Settings is for controlling which constraints will be created while creating geometry.

Sketch Curve Tool Bar

Profile is for creating a series of connected lines and/or arcs.

Line is for creating lines.

Arc is for creating arcs.

Circle is for creating circles.

Derived Curves is for creating lines parallel to other lines at a distance, or for creating a bisector line.

Quick Trim will trim geometry to other geometry.

Quick Extend will extend geometry to other geometry.

Rectangle will create a rectangle (four lines - two horizontal and two vertical).

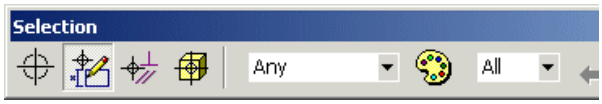
Fillet is for creating arcs that are tangent to other geometry, with or without trimming the other geometry.

Spline is for creating free form splines. This is much the same as creating splines outside the sketch. It is covered in the Curves course.

Point is for creating Points, either associative or not.

Ellipse is for creating ellipses, or general conics.

Selection Tool Bar



Select Sketch Objects is for selecting sketch geometry and sketch dimensions.

Select Constraints is for selecting geometric constraints.

Right-hand Tool Bar

Model Tree: To view your previous actions in Unigraphics you can open the model navigator. The model navigator icon is located on the navigators toolbar, and looks like this.

3D Feature Operations

These operations can all be accessed through **Insert →Feature Operation** under the **Modeling** application

Edge Blend: This operation rounds off the edge of the object

Chamfer: This operation makes an angle on the edge, similar to Edge Blend

Hollow: This operation hollows out the center of an object

Instance: This operation will repeat a procedure at regular intervals

Unite: This operation unites two solid objects

Subtract: This operation will take out the shape of one part from that solid shape of another part

Intersect: This deletes the exterior portions of the 3-D shapes, but leaves the sections that are overlapping.

3D Form Feature

These operations can all be accessed through **Insert →Form Feature** under the **Modeling** application

Extrude: This makes an object three dimensional by extending the plane out to a given distance

Revolve: This makes a 3-D object by revolving a plane a given angle around a certain axis

Sweep: This makes a 3-D part by pushing a cross section around on a given path