Rhino 7 & 8 Tools Reference Guide

1. Editing Tools

These tools help manipulate existing geometry.

- **Join**: Combines two or more curves, edges, or surfaces into one continuous object.

- **Trim**: Removes parts of objects at intersections with other objects.

- **Copy**: Creates duplicates of objects in the model.

- **Move**: Relocates objects from one place to another in the modeling space.

2. 2D Shape Creation

Tools for drawing standard 2D geometric shapes.

- **Circle**: Creates a circle based on center and radius (or other options).

- **Rectangle**: Draws a four-sided shape with right angles.

- **Polygon**: Creates an equilateral polygon with a specified number of sides.

- **Star (Polygon Star)**: Creates a star-shaped polygon with points and inner radius.

Draw some pictures to help you remember!

Notes:

3. Basic Drawing Tools

Used to draw curves and basic linework.

- **InterCrv (InterpCrv)**: Creates a smooth curve through selected points using interpolation.

- **Polyline**: Draws a series of connected straight-line segments.

- **Line**: Draws a single straight line between two points.

- **Sketch**: Freely draws curves using a digital sketching approach, useful for concept modeling.

4. Creating Curves Along Surfaces

Used to generate complex surface or solid geometry from base curves.

- **Sweep 1**: Creates a surface by sweeping a profile curve along a single rail curve.

- **Sweep 2**: Sweeps a profile curve between two rail curves.

- **Loft**: Creates a surface through two or more profile curves.

- **Pipe / MultiPipe**:

  - **Pipe**: Creates a tubular surface around a curve.

  - **MultiPipe**: Builds smooth transitions between multiple curves, ideal for organic junctions.

5. Boolean Tools

Used for combining or subtracting solid objects.

- **BooleanUnion**: Merges two or more solids into one.

- **BooleanDifference**: Subtracts one or more solids from another.

Draw some pictures to help you remember!

Notes:

Lamp Instructions:

Step-by-Step Instructions

1. **Draw the Base Line:**

   ○ Begin by drawing a guiding line where your loft will follow.

2. **Create Profile Shapes:**

   ○ Create the shapes you want to loft between (e.g., rectangle, circle, polygon).

   ○ Adjust size, orientation, and position as needed.

3. **Loft the Shapes:**

   ○ Select all the shapes in the order you want them lofted.

   ○ Type `Loft` in the command line and press Enter.

   ○ Follow the prompt to create a surface from the selected profiles.

4. **Solidify the Form:**

   ○ Use either of the following based on your needs:

      ■ `OffsetSrf`: Creates a hollow shell by offsetting the surface.

      ■ `Cap`: Closes open planar surfaces to create a solid form.

5. **Add a Hole (e.g., for Lamp Wire):**

   ○ Draw a small circle where the hole is needed.

   ○ Use `ExtrudeCrv` to turn the circle into a cylinder.

   ○ Position the cylinder where you want the hole to be.

   ○ Use `BooleanDifference` to subtract the cylinder from the main object.

6. **Add More Details (Optional):**

   ○ Create additional shapes as needed.

   ○ Use `BooleanUnion` to combine them with the main form.