A Complete Beginners Guide to Rhino 7 & 8

For Student Use Only

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Welcome!

Looking at this guidebook is your first step to understanding this overly complicated program. Rhino is dense, but once you learn a few essential workflows, it becomes a powerful modeling tool.

This Basic Guide will walk you through:

- 2D shape and curve creation
- Using editing and transformation tools
- Importing references and precision modeling
- Creating solid geometry
- Working with Boolean tools
- Building surface forms with Loft, Sweep, and Pipe
- A full step-by-step project: Making a lamp

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Section 1.1: Rhino First Steps

Opening a New File

When you open Rhino:

1. Go to **File > New Using Template**
2. Choose a template like **Small Objects – Inches** (recommended)

You will see all four viewports: Top, Front, Right, and Perspective.
Double-click **Top** to expand it.

Must-Have Tools

Turn on these tools immediately:

- **Gumball** – Helps move, scale, and rotate objects with visual handles
- **Osnap** – Enable: *End, Near, Point, Midpoint* for snapping to geometry

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Section 1.2: 2D Shape Creation & Basic Drawing Tools

2D Shape Tools (Geometry)

These tools help draw standard geometric forms:

- **Circle** – Define by center and radius
- **Rectangle** – Four-sided right-angle shape
- **Polygon** – Set number of sides, equidistant
- **Star (Polygon Star)** – Add inner radius and points

Basic Curve Tools

These are more freeform drawing tools:

- **Line** – A straight segment
- **Polyline** – Connected straight segments
- **InterpCrv (Interpolate Curve)** – Smooth curve through points
- **Sketch** – Freehand drawing, ideal for quick concepting

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Section 1.3: Organizing Your Layers

Layers help manage complexity. Rhino opens with layers like Black (Default), Red, Blue, Green, and Purple.

You can:

- **Lock layers** – Useful for reference images or guides
- **Change layer of an object** – Select object > Type `ChangeLayer` > Select desired layer
- **Use `Hide` and `Show`** to declutter your workspace

Tip: If you lose track of your object, use `ZoomSelected`. Select the object and click each viewport once to reorient.

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Section 1.4: Editing Tools

Use these to manipulate geometry after you draw it:

- **Join** – Combine two or more curves/surfaces
- **Trim** – Cut off parts of geometry using other shapes
- **Move / Copy** – Relocate or duplicate
- **Scale / Rotate / Mirror** – Transform shape and direction

Note: Gumball can perform most move/scale/rotate functions quickly.

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Section 1.5: Precision Modeling & Importing Images

If you're modeling for functional parts (lamps, furniture, ceramics), accuracy matters.

Use blueprints from manufacturers or your own drawings:

1. Download or screenshot a blueprint as .jpg or .png
2. Drag it into Rhino

3. Select **Picture** when prompted
4. Draw over the reference using 2D or curve tools

Tip: Move images to a separate layer and lock it to avoid accidental selection.

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Section 2.1: Extruding Curves

Once you've drawn shapes using Section 1.2 tools:

- Select the shape
- Type `ExtrudeCrv`
- Set a height
- Toggle **Solid = Yes** to make a watertight object
- If Solid is off, use Cap to close the top/bottom

This is the most direct way to turn 2D into 3D.

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Section 2.2: Boolean Operations

Booleans allow you to combine or subtract shapes:

- **BooleanUnion** – Merges solids together
- **BooleanDifference** – Subtracts one from another

Use BooleanDifference to make holes, slots, or inlays.
Make sure your shapes are solid before using Boolean tools.

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Section 2.3: Giving Objects Mass

Extruded objects may look 3D but still be surface-only. Use `OffsetSrf` to give them wall thickness:

- Select your surface
- Type `OffsetSrf`
- Set a thickness
- Choose direction (inward or outward)
- Confirm **Solid = Yes**

This is crucial when preparing models for 3D printing or physical production.

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Section 3.1: Loft 101

Loft is a flexible way to connect profiles into a form.

Steps:

1. Draw multiple shapes (circles, rectangles, etc.)
2. Use **Gumball** (blue arrow) to raise shapes in the Z-axis
3. Select shapes in order (hold Shift if needed)
4. Type `Loft`
5. Choose a style: *Loose, Uniform, Tight*
6. Press Enter to confirm

Loft creates a surface stretched between curves.

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Section 3.2: Other Surface Tools – Sweep & Pipe

Sweep 1

- Uses 1 rail and 1 profile
- Good for handles, pipes, cords
- Draw profile (circle/rectangle)
- Draw rail (line, polyline, etc.)
- Type Sweep1

Sweep 2

- Uses 2 rails and 1 profile
- More control over shape twist and volume
- Draw profile and two unjoined rails
- Type Sweep2

Pipe / MultiPipe

- **Pipe** – Add thickness to any curve
- MultiPipe – Blends multiple curves together organically

Use Line, Polyline, or Sketch tools to create your paths.

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Section 3.3: Making a Lamp with Loft – Step-by-Step

1. **Draw the Base Line**
    - A vertical curve as the spine
2. **Create Profile Shapes**
    - Circles or polygons at intervals along the spine
    - Move upward with Gumball (Z-axis)
3. **Loft the Shapes**
    - Select all shapes > Type `Loft` > Adjust settings
4. **Solidify the Form**
    - Use `OffsetSrf` for wall thickness
    - Or use `Cap` to close ends
5. **Add a Wire Hole**
    - Draw a small circle
    - Use `ExtrudeCrv` to make a cylinder
    - Position it where the wire goes
    - Use `BooleanDifference` to subtract
6. **Add More Details (Optional)**
    - Draw additional forms
    - Use `BooleanUnion` to attach

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Section 4.1: Sweep Tool Review

Sweep 1

- Profile: e.g. circle
- Rail: e.g. wavy curve
- Result: Object follows single path

Sweep 2

- One profile
- Two rails (e.g. mirrored curves)
- Result: More complex surface control

Both tools work well for organic shapes like jewelry, handles, and sculpture.

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Final Notes

- Save frequently
- Use Undo (Ctrl + Z) often
- Use Layers and Zoom tools to stay organized
- Most tools can be typed in directly or found in the top ribbon
- The Rhino command line is your best friend—watch what it asks for