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Jewellery Design in Rhino 8: A Technical & Creative Workflow



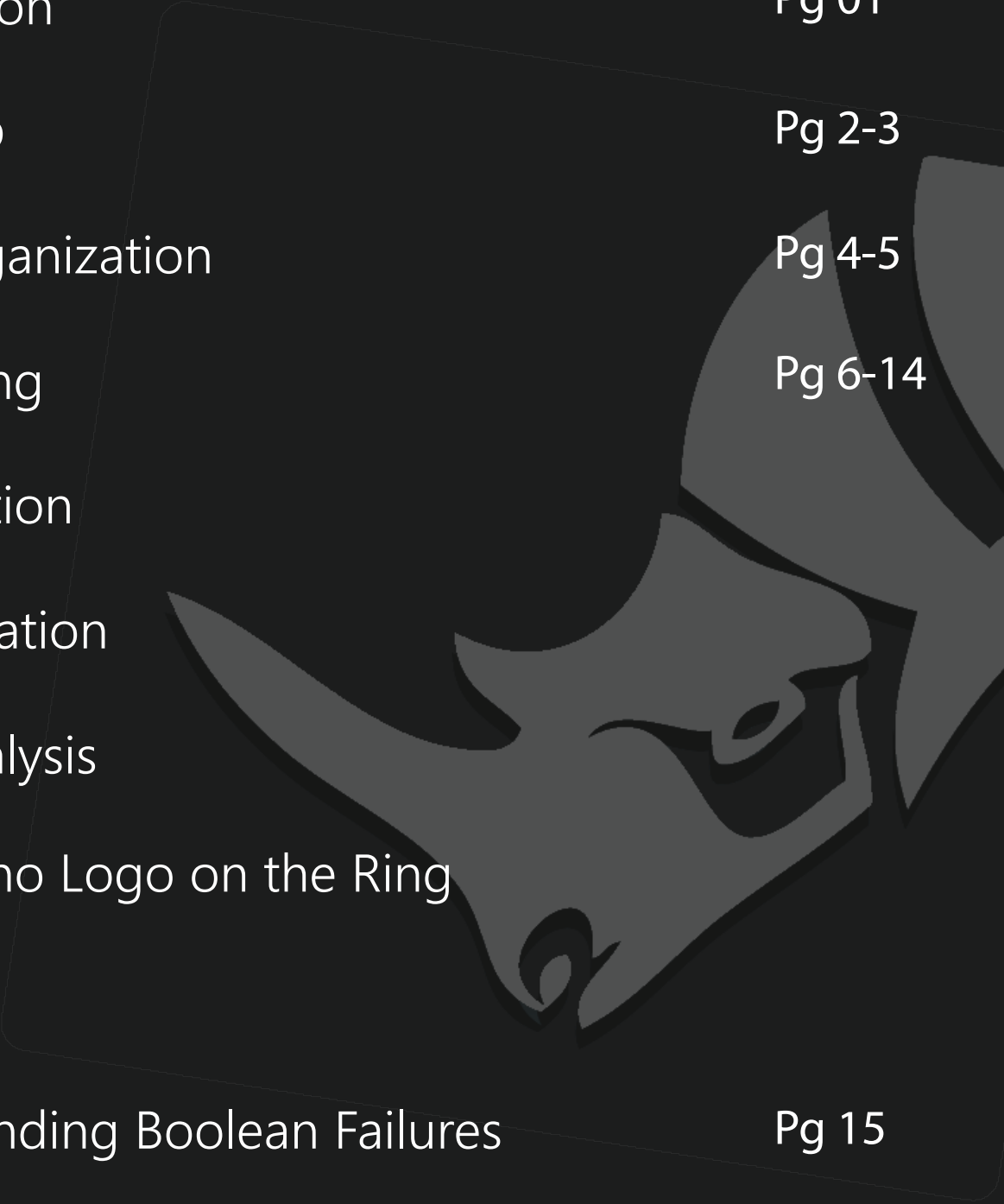
Concept & Creation by:

VANSHI MANDALIYA



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1. Introduction

About Me:

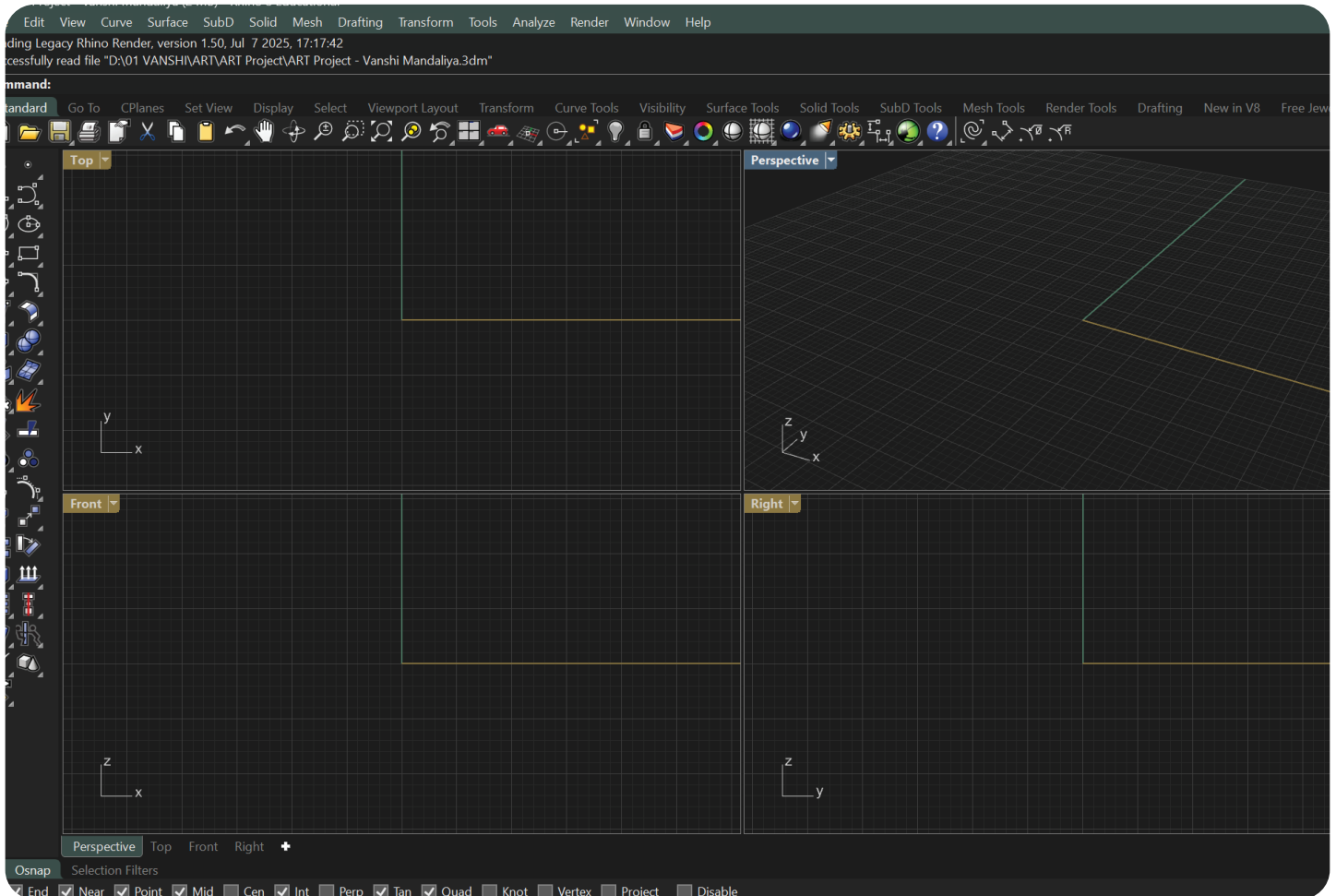
I am Vanshi Mandaliya, a jewelry CAD designer and educator from India, and the proud 7th generation in my family's jewelry legacy. Started early at the age of 15, Skilled in Rhino 3D, digital design, and manual artistry, I combine heritage with innovation to create meaningful designs. My passion lies in teaching and guiding aspiring designers to develop both their creative vision and technical expertise.

Project Goals:

- Teach essential Rhino 3D tools and workflows for jewelry design.
- Demonstrate precision techniques for clean curves and surfaces.
- Share efficient methods for organizing layers and tools.
- Guide in creating realistic jewelry renders for presentation.
- Provide practical tips to enhance both creativity and technical skills.

2. File Setup in Rhino

1) Setting up the User Interface



1. What's the Best Way to Customize Your UI?

-> Units & Template

- For jewellery design modelling, always work in millimeters unit.
- Check units via File - Properties - Units, before modeling.
- For template, Start with Small Objects – Millimeters template to avoid tolerance issues.

(Remarks: Set Absolute Tolerance: 0.001 mm and Angle Tolerance to 1° in properties - unit for smooth curves & jewellery precision.)

-> Customising the User Interface

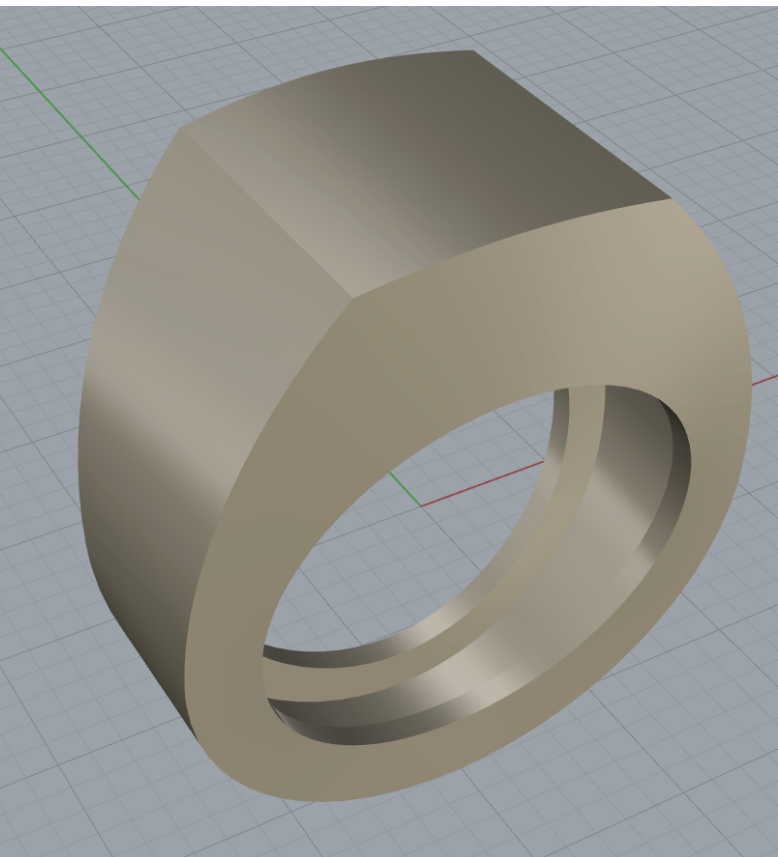
- As you can see above, I've customized the UI - you can do the same by following the steps mentioned below:
- > Go to Tools - Options - [Appearance](#), and choose a UI color theme that suits your style.

2. Why Customize the Appearance?

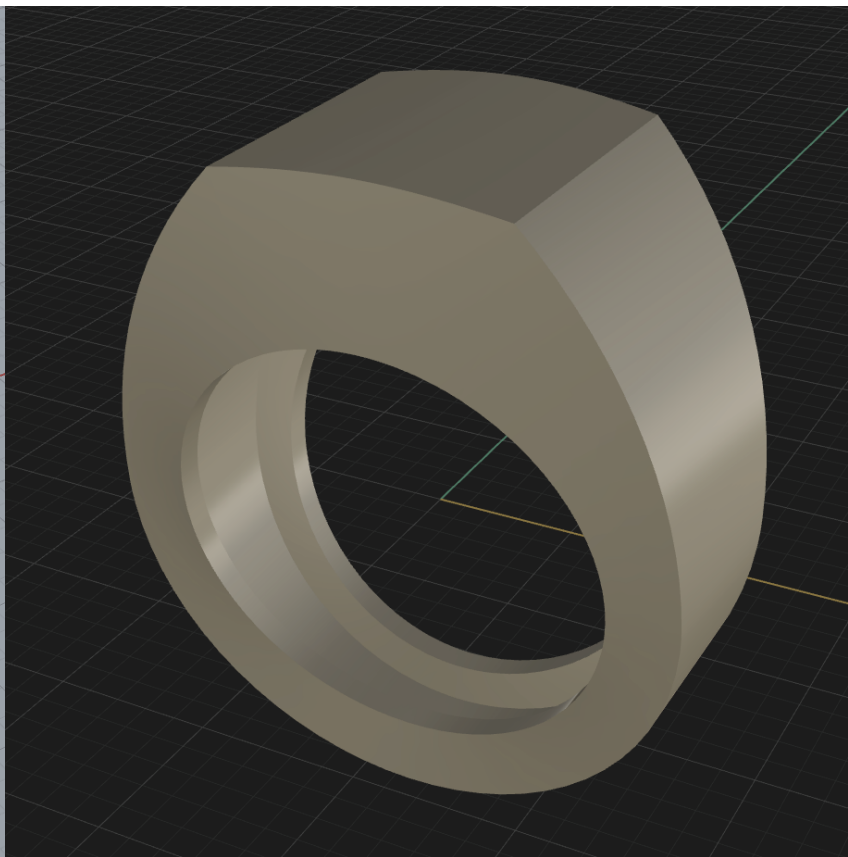
-> Points you must consider

- The right UI setup ensures metal and gem details are crystal clear, and Rhino gives you the customization options to achieve it.
- Reduces eye strain during long work sessions.
- Makes key elements stand out.
- Matches your personal workflow style.

Before - Workspace



After - Workspace



Guess what: A well-arranged UI can save up to 20% of your working time.

3. Layer Organization

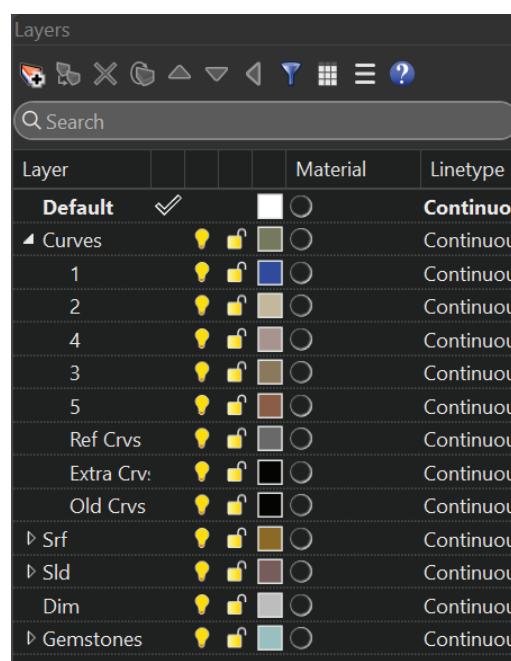
1. Why Layer Organization Matters?

- > • Keeps different parts of your design (band, stones, settings, prongs) separate.
- Makes editing faster without affecting other components.
- Improves file readability for you and anyone else who works on it.

2. How to organize layers?

-> Layer Structure I Prefer:

- Curves (Parent Layer) – Child layers named for specific curve sets.
- Surfaces (Parent Layer) – Child layers for individual surface parts.
- Solids (Parent Layer) – Child layers for each solid component.
- Gems (Parent Layer) – Child layers for center stones, side stones, and accents.
- Dimensions (Parent Layer) – Child layers for measurement annotations.
- I also recommend adding Extra and Ref. child layers under each parent layer.
 - Extra -> For temporary geometry that you may still need later.
 - Ref. -> For reference geometry or imported guides that assist in modeling.
- This setup keeps my workflow organized, speeds up editing, and reduces the risk of errors during design and production.



(Refer page no. 05 for steps of creating layers)

3. Creating and Managing Layers

-> Creating **Layer** & adding color:

1. To Create a Layer:

Open the Layers panel - click New Layer icon, to add a new layer. 

2. To Add Color:

Right-click the layer - Change Object Color - choose your desired color. 

3. To Create a Sublayer:

Right-click the main layer - New Sublayer - name it accordingly or you can also use the sub-layer icon to create one. 

4. To Rename:

Double-click any layer or sublayer name to rename it or you can right click and select Rename.

5. Set Active Layer:

Double-click a layer name to make it active - new objects will be created on this layer.



6. Lock/Hide Layers:

Use the lock or bulb icons in the Layers panel to protect or hide objects while working.



4. Saving your Customised Template

-> How to save the customised template you created:

1. Saving a Template in Rhino

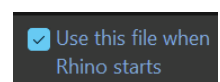
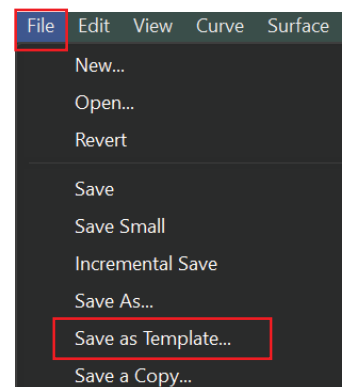
2. Set up your document with preferred units, layers, colors, and settings.

3. Go to File - **Save As Template...**

4. Choose a name and location (default template folder is recommended).

5. Click OK - your custom template will now appear when starting a new file.

6. When opening a new file (File -> New), check the "Use this file when Rhino starts" option so that your customized template automatically loads every time Rhino opens.



E.g.: Save your layer structure for reuse: File -> Save As Template -> Name it 'Jewellery Layer Template'.

4. Signet Ring

1. Starting with the curves

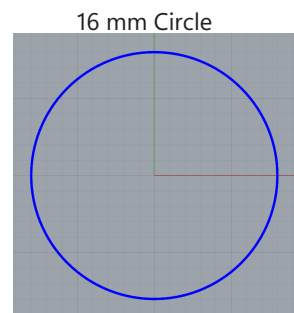
-> Getting Started with the Template File

- Open the **Signet Ring Template File**.
 - In the template, hide the completed ring in the '**DONE**' hierarchical layer.
 - The unit of the model is in **Millimeters (mm)**.
- Now you are ready to begin!

1) Starting with creating Inner and Outer Diameter of the ring

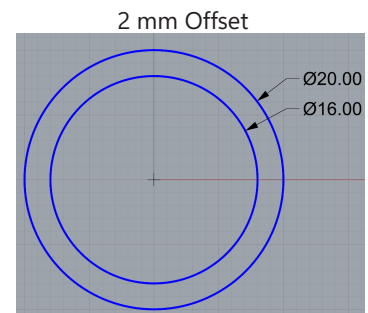
-> Start with a Circle

Make a 16 mm (Diameter) **Circle**: center, radius  from '0' origin in FRONT view. (Make sure to use the **dimension** tool  to cross check the circle's diameter.)

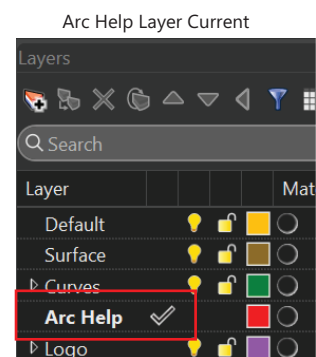


-> Create Offset Curve

Use **Offset** curve  command to create parallel circle outside of the 16 mm circle and offset distance will be 2 mm.

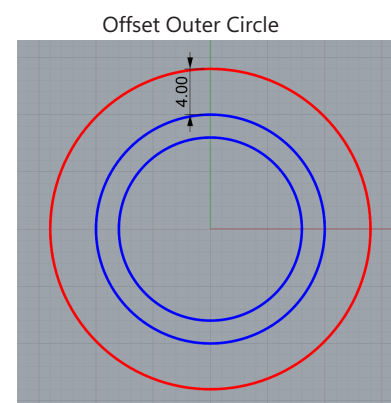


-> Make the 'Arc Help' layer current



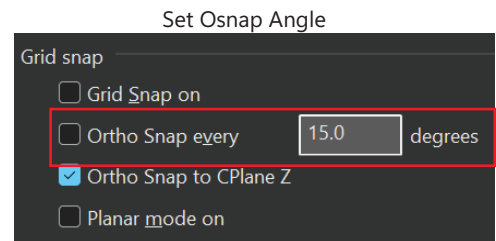
-> Offset a Curve by 4mm

- Select the outer curve to create an offset curve.
- Type `_Offset` in the command line and press Enter.
- At the command prompt:
Specify the distance = 4mm.
- Click on the outside of the curve.
- A new curve appears, parallel to the original, on the Arc Help layer.



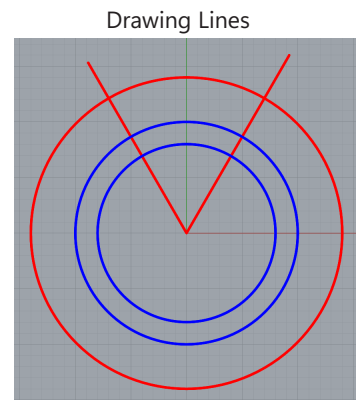
-> Set **Ortho** Angle to 15°

- Look at the status bar at the bottom of Rhino.
- Make sure Ortho is ON (highlighted).
- Right-click the Ortho button - choose Settings.
- In the dialog box, set Ortho angle = 15°.
- Click OK.
- Now any lines you draw with Ortho ON will snap to multiples of 15°.



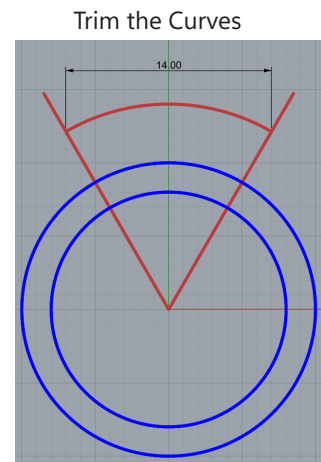
-> Draw Guide Lines

- Type **_Line** and press Enter.
- Start from the '0' origin.
- Draw a line using the Ortho snap at 30° (you'll see them lock into that angle).
- Click at 30° and place the line.
- **Mirror** the line to the other side using mirror command.



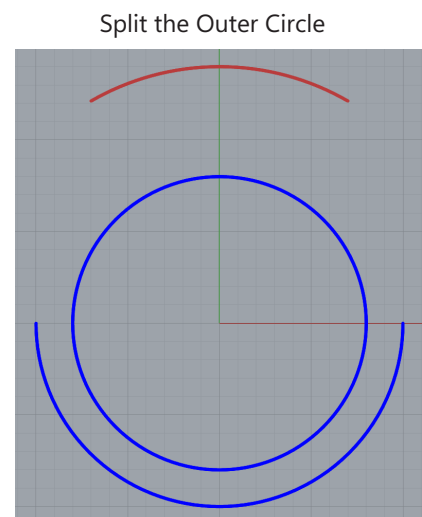
-> Trim or Create the Arc

- Use the **_Trim** command.
- Select the two lines as cutting objects.
- Trim away the extra parts of the offset curve, leaving only the arc segment at the top.
- Hide the guide lines, after trimming.



-> Split the Outer Circle

- Now we'll split the outer circle (the one we offset earlier by 2mm) into half.
- Type **Split** in the command line (or choose it from the toolbar).
- When prompted, select the circle.
- In the command line, click Point option. **Select cutting objects (Point):**
- Turn on Osnap and snap from Quad to Quad along the X-axis in Front View.
- The circle will split into two halves.
- Delete the upper arc.



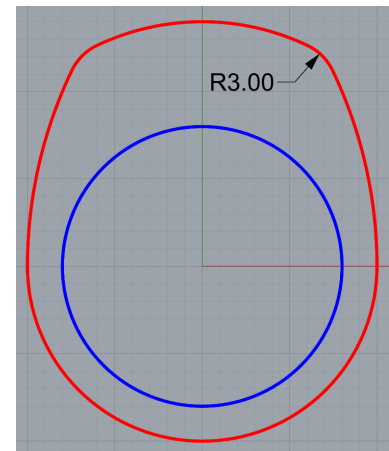
-> Using Extend Command

- Type **Extend** in the command line.
- When Rhino asks to select boundary objects, press Enter.
- Turn on Osnap - End to snap easily.
- Select the lower arc (the one that was split earlier)
- Then Select 'ToPoint' option in the command-line then, snap to the end of the upper arc.

End of extension or enter extension length <3.238> (Center **ToPoint**):

- Repeat the same on the other side.
- Now Join the arcs together.
- Finally, use **Fillet** with Radius = 3 to round the corners.

Complete the Outer Curve

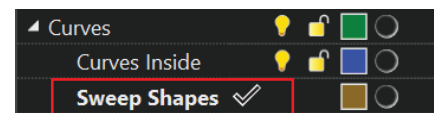


2) Now we'll create Sweep Curves for the ring

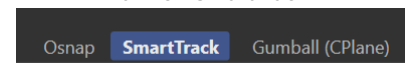
- Make the '**Sweep Shapes**' layer current

-> Turn on **SmartTrack**

- Look at the status bar at the bottom of Rhino.
- Make sure the SmartTrack button is highlighted (ON).
- If it's not, simply click it to activate.
- You can also type `_SmartTrack` in the command line and press Enter.
- Turn ON End, Mid, and Near Osnap in the status bar



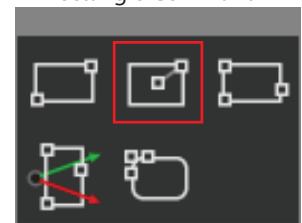
Turn on SmartTrack



-> Activate the Rectangle Command

- Type `_Rectangle` in the command line, or
- Go to the Curve menu - Rectangle: Center, Corner option.
- Rhino will now ask you to specify the center of the rectangle.

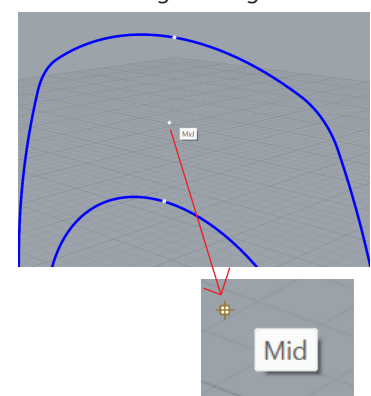
Rectangle Command



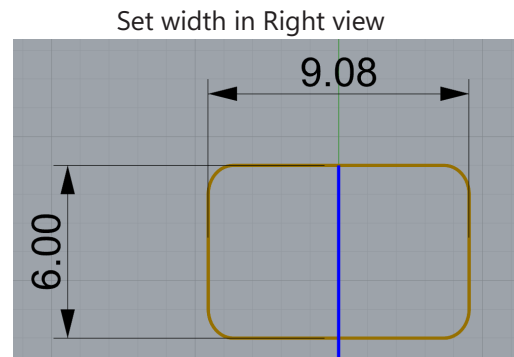
-> Place the Center of the Rectangle with SmartTrack

- Hover your cursor at the midpoint of the outer curve.
- Hover your cursor at the midpoint of the inner curve.
- Wait until Rhino displays a small white smart point.
 - If the smart point doesn't appear, press Ctrl to make it visible and to remove it, press Ctrl again.
- Place your cursor at the intersection point of the two SmartTrack markers (the midpoint), then start drawing the rectangle from there.
- When the SmartTrack midpoint appears, click on it to place the rectangle's center.

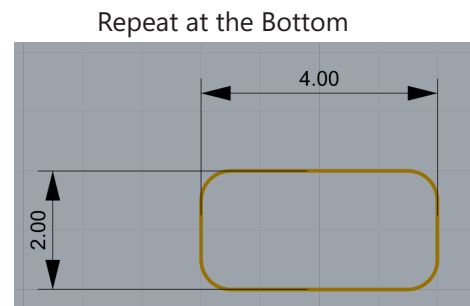
Creating Rectangle



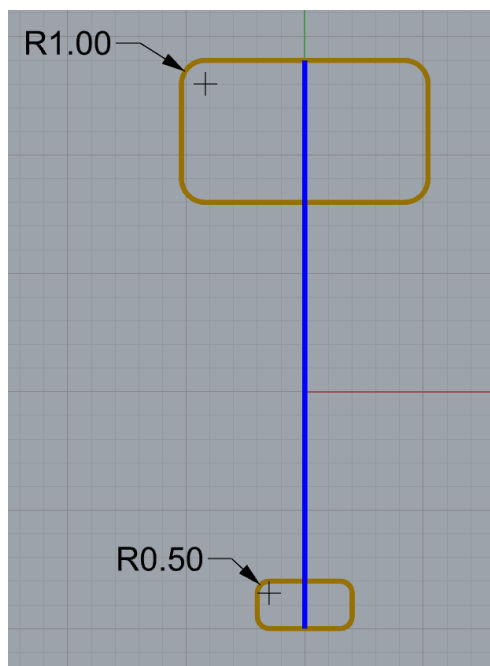
- > • Now switch to the Right View. Adjust the rectangle's width until the total size measures around 9 mm. The final rectangle should be approximately 9 × 6 mm.
- Apply Fillet with radius 1 mm to make rounded corners.



- > Repeat the same process at the bottom to create a small sweep shape with a rectangle size = 4x2 mm using smart track, and fillet it to 0.5 mm radius.



- > Here's how the Sweep Shapes will look in right view:



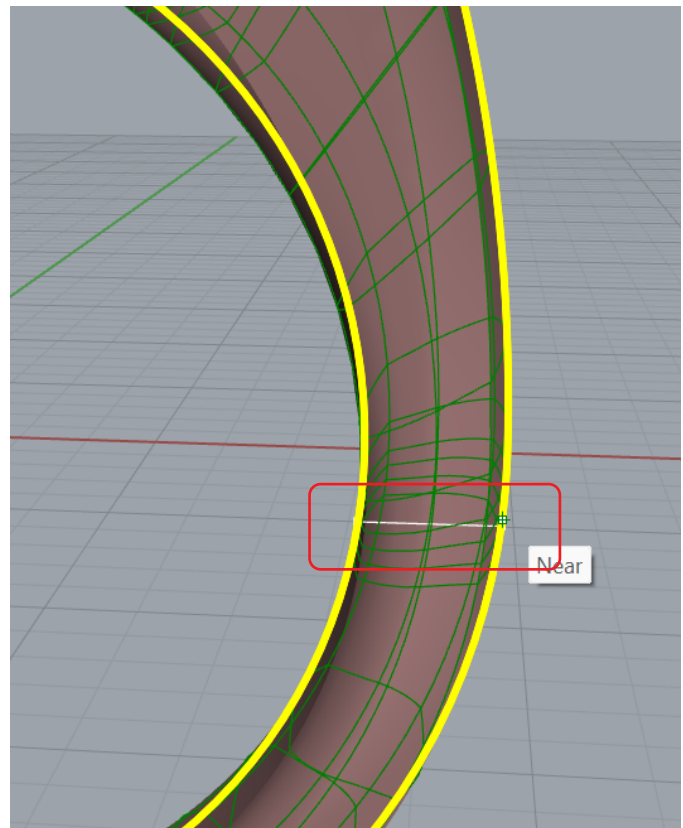
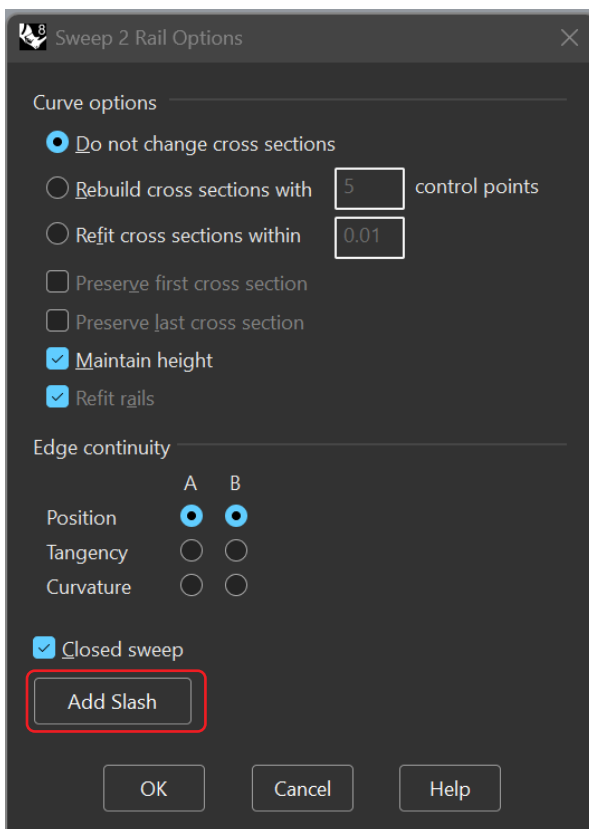
2. Onto Surface Creation & Analysis

3) Next step is to use our curves to create a clean, continuous surface

- Make the '**Surface**' layer current



- > • Start the command
Surface - **Sweep 2 Rails** (or type Sweep2).
- Select the rails
Click the two ring curves as Rail 1 and Rail 2.
- Select the profiles
Click the top and bottom rectangles as the profile curves.
- Align seam points
In the preview, drag the seam arrows so both profiles point in the same direction on along rail.
Reason: Misaligned seams cause twisted/corkscrew surfaces.
- Use "Add slash"
Click Add slash, then place a matching slash on both rails at the same location. Repeat if needed.
Reason: Adds additional cross-section alignments to control how the surface is created between sections.



Using Add Slash

(See the next page for surface creation visuals)

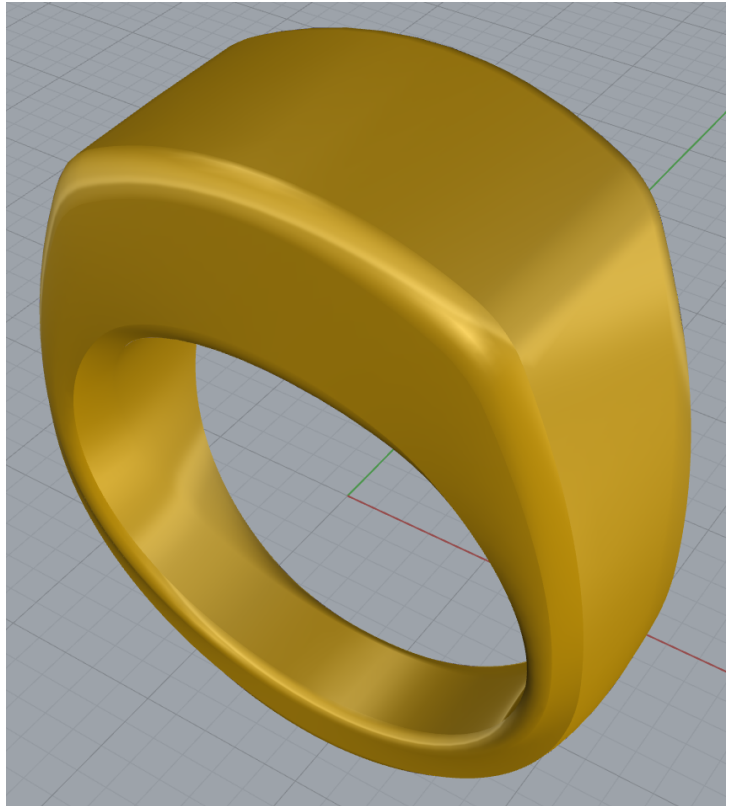
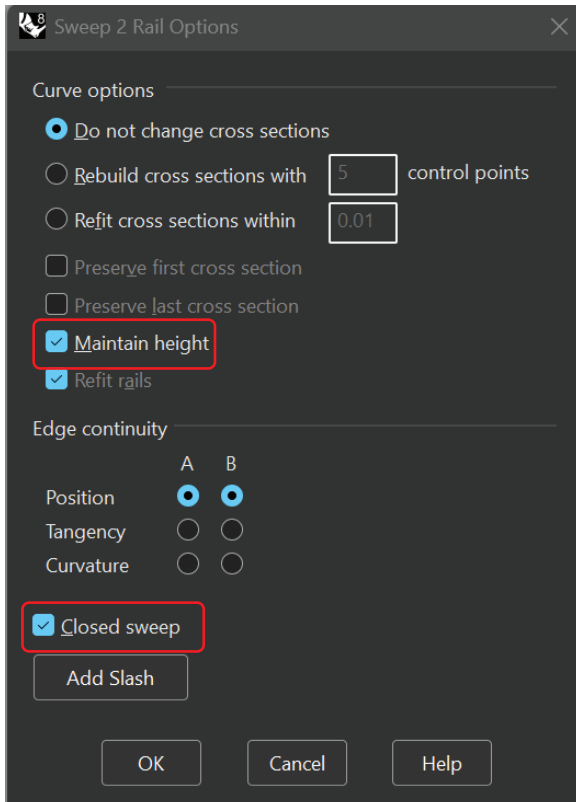
-> • Set options

In the dialog, check:

- > Maintain Height - Keeps the profile height consistent.
- > Closed Sweep - Use if making a full ring around (360°).

• Build the surface

Press Enter/OK to complete the sweep.

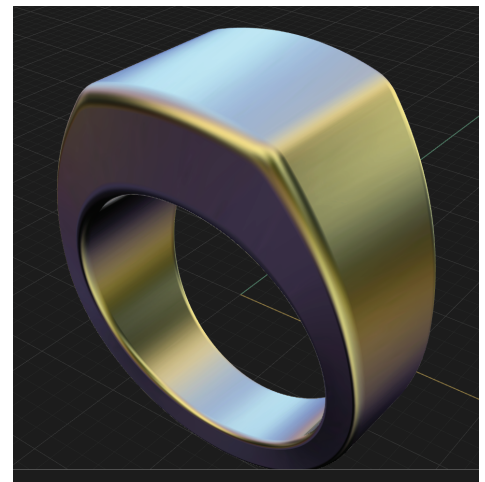


Creating the Srf

4) Let's Analyze the Surface!

- > • Your ring is almost ready - but before moving on, check the surface for smoothness and accuracy.
- Use Rhino's Analysis tools, such as [Zebra](#) [Environment Map](#) (Emap), to inspect surface flow and reflections.
- In this example, I've used the Brushed Silver Emap.
- Look closely at the reflection patterns:
Smooth, even stripes = clean surface.
Wavy or broken patterns = bumps or mismatches in your curves.
- If you find any distortions, go back and correct the curves or rebuild the surface before starting the detailing stage.

Analyzing the Surface



Let's Add Rhino Logo on the Ring!

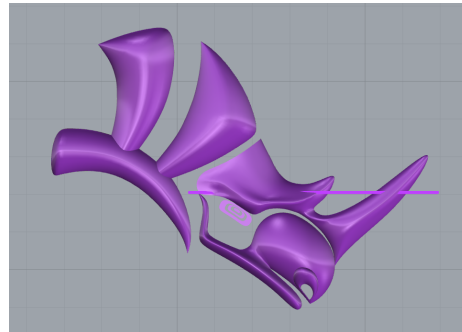
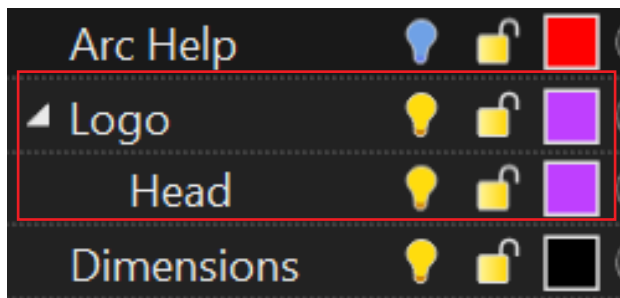
-> Turn On Layer

Make sure the Rhino Logo srf layer is visible

(logo surface is already prepared there along with ref. line).

Make sure the History command is on.

Turn on Rhino Layer



-> Command

Type **OrientOnSrf** in the command line.

-> Select Logo

Click the Rhino logo surface and press Enter (Avoid selecting the reference line).

-> Set Base Point

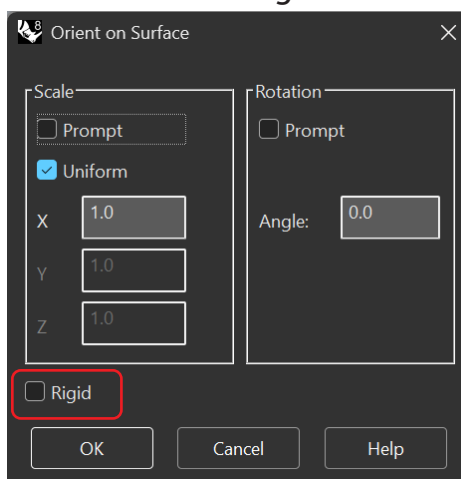
Pick the base point of the logo using the reference line provided from one end to another end.

-> Surface to Orient On

After this, a dialog box will appear → uncheck the Rigid option so the logo can flow naturally on the curved surface.

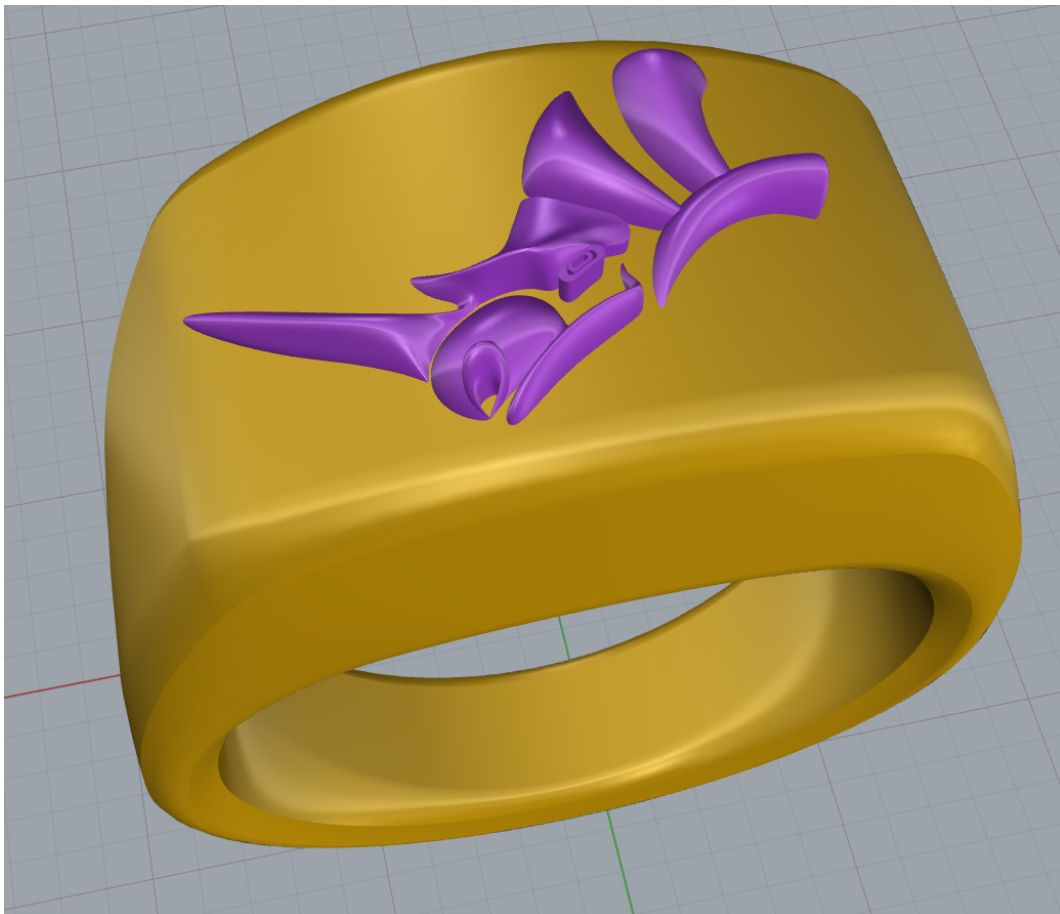
(In the command line, set Copy = Yes (so the original logo stays intact on the layer).

Orient Dialog Box



- > Point on Surface to Orient on
Click on the top of the ring surface to position the logo and then press Enter.
- > Adjust Original Logo
Since history is on, you can edit or move the original Rhino logo srf and the oriented (child) logo on the ring will update automatically.

Here's how the final result will look:



Orient On Srf

3. Rendering & Presentation

4) Now we'll render the Signet Ring for final presentation

-> Steps for Rendering the Ring:

- Switch to Rendered display mode.
- Apply Metal material from the [Materials](#) panel.
- Add lighting or use an HDRI environment.
- Set a clean background.
- Position the model nicely.
- Check render settings.

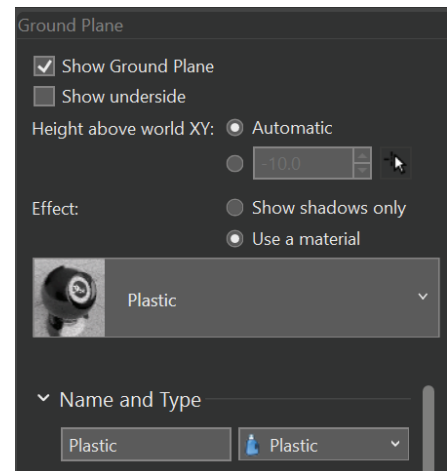
Applying Material



-> Steps for Setting the Ground Plane:

- Go to [Ground Plane](#) Panel.
- Under Effect, select Use material.
- Select Import from Material Library.
- Then, select the material you want as a background.
- I added the Plastic - Black color material, you can play around with the render settings to find what works best for your design.

Adding ground plane



4. Here is the final result!

-> **Save Final Ring Image**

1. In the Perspective viewport, set display mode to Rendered.
2. Adjust the view (rotate, pan, zoom) to a good angle.
3. Type [ViewCaptureToFile](#) in the command line → press Enter.
4. In the dialog box:
 - Choose file name & location.
 - Select PNG or JPG.
 - Set resolution (e.g., 1920 × 1080).
 - (Optional) Tick Transparent Background.
5. Click Save.



5. Understanding Boolean Failures

1. Why Boolean Commands Fail in Rhino?

Note these 5 common reasons causing **Boolean union** to fail in 3D modeling:

- > 1. Non-Intersecting Objects
 - The objects do not physically intersect or touch each other.
 - Boolean Union requires some overlapping geometry to combine the volumes.
- 2. Open or Invalid Geometry
 - One or both objects are not closed solids (they have holes, naked edges, or bad joins).
 - Boolean operations require valid, watertight solids to work correctly.
- 3. Coplanar or Overlapping Faces
 - Objects share exactly overlapping faces or edges, causing the Boolean to fail.
- 4. Tiny Gaps or Tolerances Issues
 - There are small gaps or misalignments between the objects, often due to tolerance settings.
 - The software may not recognize the objects as touching or intersecting.
- 5. Self-Intersecting Geometry
 - One or both objects have self-intersecting surfaces (like a twisted or folded surface).

2. Quick Boolean Checklist

- Before using Boolean, ask yourself:
 - > 1. Are both objects closed solids?
 - 2. Do they intersect properly?
 - 3. Are there any naked edges or gaps?
 - 4. Is the geometry unnecessarily complex?

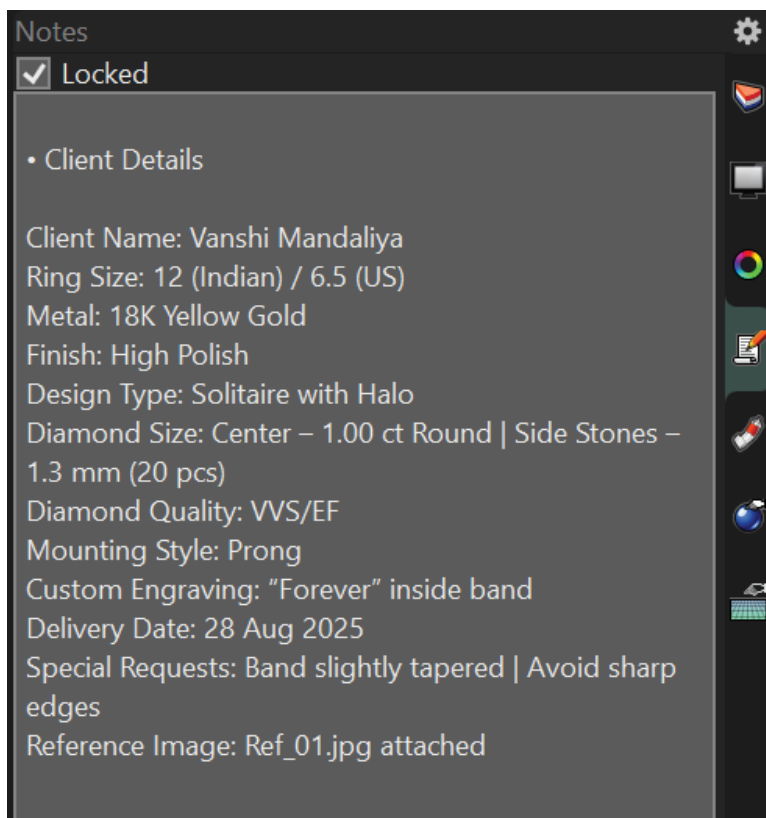
Guess what: Always keep a copy of your model on a separate "Master" layer, it's a smart backup for future edits or changes.

6. Quick Tips for Smooth Workflow

1. Using Notes Panel

- Adding a [notes](#) panel in Rhino is a quick yet powerful way to keep track of essential project details. It helps ensure you don't miss client requirements or design specifics, and it's as easy as typing directly into the Notes tab! This is something I personally use all the time to streamline the process.

-> Check the image below to understand it better:



2. Easy Access to Commands

- Just like popup toolbars and aliases make it easy to access commands, here's another helpful method worth knowing, you can create your own custom "Go-To" toolbar for the commands you use most often.

-> The following method of creating custom 'Go-To' toolbar adds another level of ease to your workflow:

- Right click on the standard toolbar -> select 'New Toolbar' to create one and rename & add your go-to commands for easy workflow.

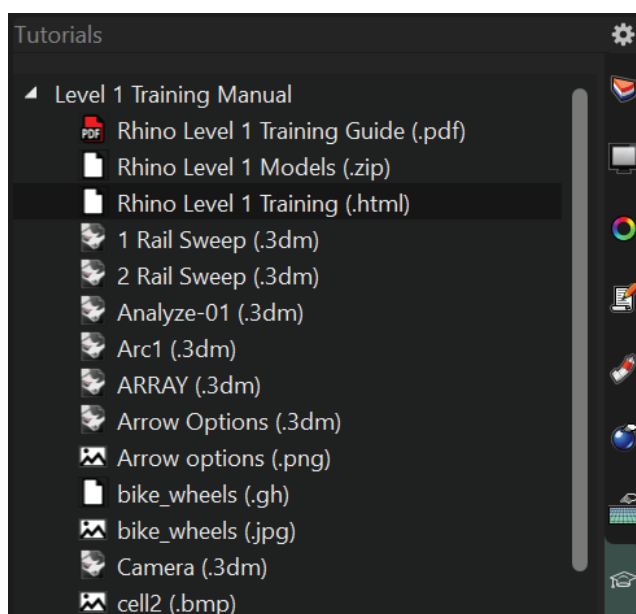


3. Smart Moves in Rhino

- Must-Follow Tips to Excel in Rhino, Work Faster & Achieve Better Results:
 - > • Always start your design from center.
 - Rebuild curves at every step to achieve smoother surface.
 - Rebuild offset curves to reduce control points and keep the geometry clean.
 - Layer your objects step by step for better clarity and control.
 - Keep the Gumball active for quick edits.
 - Use Analyze tools to check surface quality before proceeding.
 - Save incrementally to avoid losing progress.
 - Use the keyboard for commands and aliases, it speeds up your workflow significantly.

4. Use Rhino Tutorials for better practice

- Practice modeling using Rhino models - as it builds speed, confidence, and deeper command understanding.
 - > • Go to [Tutorials](#) panel.
 - Choose a specific command or model to practice with.
 - Try resolving errors on your own first, it builds confidence and problem-solving skills.
- > Check the following image to find Rhino Models:



Thank You for Being a Part of This Journey

This project was more than just modeling a ring - it was about understanding the workflow, organizing thoughtfully, and learning to approach each step with intention. From setting up layers to analyzing surfaces and refining details, every action contributed to a cleaner, smarter design process. Rhino becomes more intuitive the more you explore it. The key is to stay curious, experiment often, and not fear mistakes - they're often your best teachers.

This is just one design. There's always more to create, more to refine, and more to learn.



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