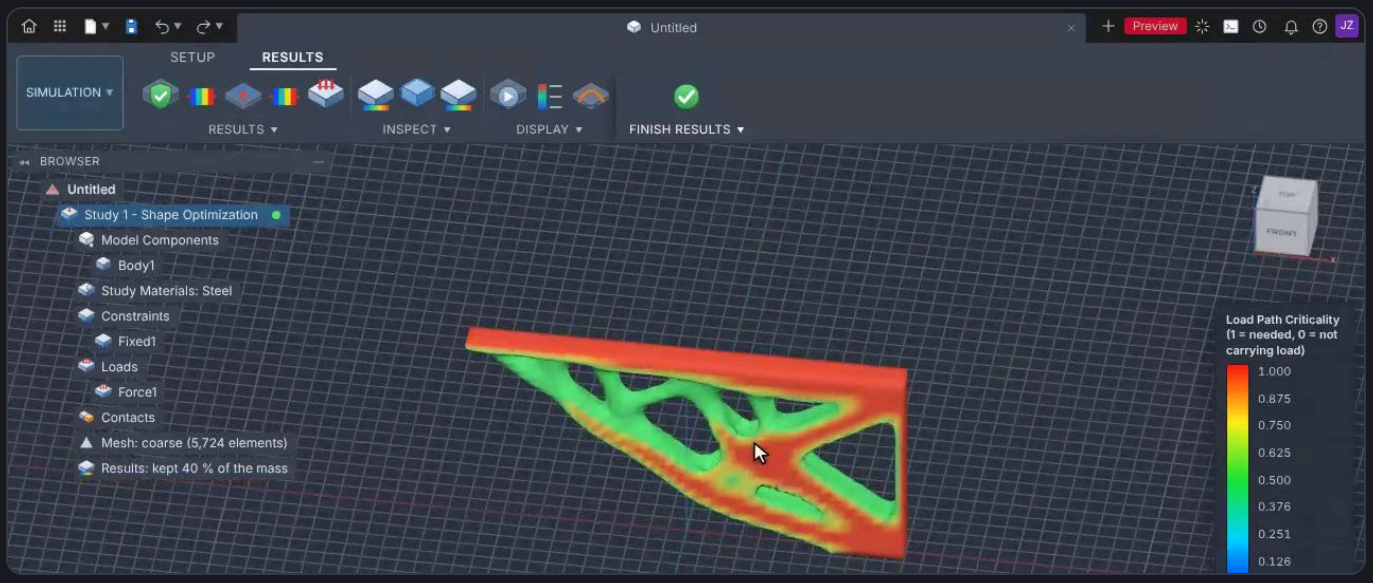
CIPHERFORGEDSTUDIOS

USER GUIDE

Ironcast

Parametric 3D CAD for Windows



For Ironcast 0.19.1 (beta)

59 lessons · 12 chapters

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START

Welcome to Ironcast

Ironcast is a parametric 3D CAD program for Windows. You sketch a shape, turn it into a solid, and keep shaping it. Every step you take is kept in a timeline, so you can go back, change a number, and the rest of the model updates.

It covers a lot of ground in one program: 3D modeling, assemblies, drawings, 2D drafting with DWG files, stress simulation, toolpaths for machining, sheet metal, circuit boards, and ladder diagrams with a PLC simulator for control panels.

This guide has two parts. The first few pages explain the screen, the mouse and the basics of files. After that come 59 short lessons, grouped by subject. Each lesson is also a narrated video inside Ironcast, so you can read, watch, or do both.

> **Ironcast is in beta.** It is used every day and updated often, and some rough edges remain. Check important measurements and simulation results yourself before you rely on them.

Installing

1. Download the zip file from the Ironcast page.
2. Open your Downloads folder, right-click the zip and choose **Extract All**.
3. Open the extracted folder and run the Ironcast setup file.
4. If Windows says “Windows protected your PC”, click **More info**, then **Run anyway**. This shows for new programs that are not yet signed with a paid certificate.

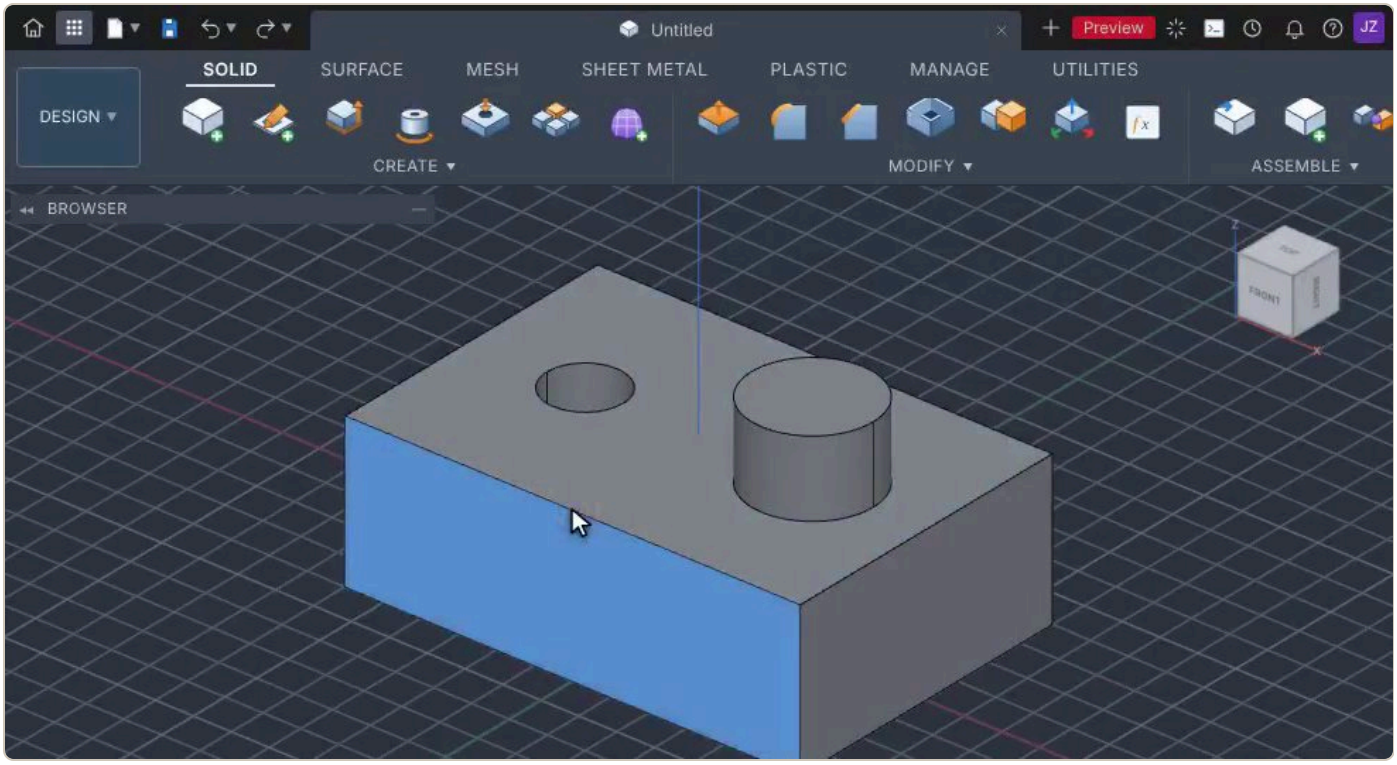
You need Windows 10 or 11 (64-bit), about 600 MB of free disk space and a graphics card from roughly the last ten years.

Updates

Ironcast checks for a new version when it starts. When one is ready it tells you, and installs it when you restart. Updates are checked against a digital signature before anything is installed. You can also check by hand: click the **?** button at the top right and choose **Check for Updates**.

START

The screen

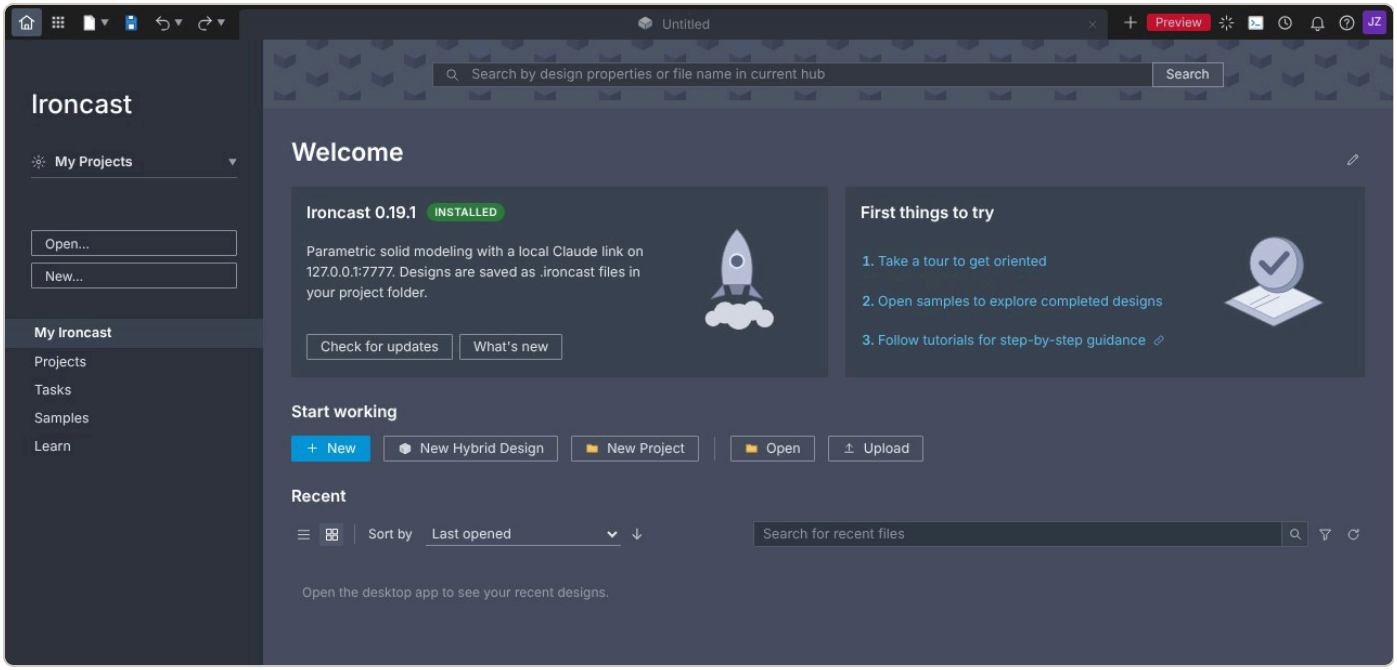


Top bar	Open and save files, Undo and Redo, and a tab for each open design. On the right are notifications, the **?** Help menu and your user menu (theme, Preferences).
Workspace button	The big button at the left of the toolbar (it says **DESIGN**). Click it to switch to another workspace, such as Simulation or Drawing.
Toolbar	Tabs (Solid, Surface, Mesh, Sheet Metal, Plastic, Manage, Utilities) and, under them, groups of tools such as Create and Modify. The small arrow under each group opens a menu with every tool in it. On a narrow window some buttons fold into those menus.
Browser	The list on the left: your bodies, sketches, planes and components. Click the eye to hide or show something. Right-click for more.
Canvas	The big area in the middle where your model lives.
ViewCube	Top right of the canvas. Click a face to look straight at that side, or the little house to go back to the home view.
Navigation bar	Bottom middle: orbit, pan, zoom, Fit, display settings and Viewports (two, three or four views at once).

Timeline	Along the bottom: one icon for every step you have made. Double-click a step to change it. Drag the marker back to see the model as it was.
Data panel	Slides out from the left (the grid button at the top left). It shows your design files and folders.

The Home screen

The house button at the top left opens Home. From there you can start a new design, open a recent one, and open the **Learn** tab, which holds all the video lessons.



START

Mouse and keyboard

Mouse

Pan (slide the view)	**Hold the mouse wheel down and drag**
Orbit (spin the view)	**Shift + hold the mouse wheel and drag**
Zoom	**Roll the mouse wheel (zooms toward the pointer)**
Select	**Left-click (Shift or Ctrl adds to the selection)**
Window select	**Drag from left to right**
Crossing select	**Drag from right to left**

Select a loop of edges	**Double-click an edge**
Marking menu	**Right-click**

While modeling

Search for a command	**S**
Extrude	**E**
Press Pull	**Q**
Fillet	**F**
Hole	**H**
Move/Copy	**M**
Measure	**I**
Joint	**J**
Show or hide	**V**
Scripts and Add-Ins	**Shift+S**
Window / Freeform / Paint select	**1 / 2 / 3**
Fit the model in the view	**F6**
Undo / Redo	**Ctrl+Z / Ctrl+Y**
Save / Save As	**Ctrl+S / Ctrl+Shift+S**
Open / New	**Ctrl+O / Ctrl+N**
Delete	**Del**
OK / Cancel a dialog	**Enter / Esc**

While sketching

Line	**L**
2-Point Rectangle	**R**
Center Diameter Circle	**C**
Sketch Dimension	**D**

Switch a line to construction	**X**
Type a size while drawing	**Type the number; Tab moves to the next value**
End a chain of lines	**Double-click, or Esc**

The same list is in Ironcast under **?** › Keyboard Shortcuts.

START

Workspaces

A workspace is a set of tools for one kind of job. Click the workspace button at the left of the toolbar to switch.

Design	3D modeling: sketches, solids, surfaces, meshes, sheet metal and assemblies. This is where you start.
Generative Design	Ironcast works out shapes for you from the parts that must stay, the space to keep clear, and the loads. See page 74.
Render	Appearances, lighting and finished pictures of your design.
Animation	Exploded views and storyboards that show how things go together.
Electronics	Schematics and circuit boards, with a 3D view of the board.
Panel and Control Panel	Electrical control work: ladder diagrams, a PLC simulator, motor circuits and panel layout.
Simulation	Stress, vibration, heat and more, solved on your own computer.
Manufacture	Toolpaths for milling and routing, a cut preview and G-code.
2D Drafting	AutoCAD-style 2D drawing with typed commands. Opens and saves DWG and DXF.
Drawing	Dimensioned sheets made from your 3D model.

START

Files, saving and sharing

A design is saved as one **.ironcast** file. Press **Ctrl+S** to save. Your designs stay on your computer: Ironcast has no account and does not send your files anywhere.

Getting a design out

3D printing	STL or 3MF. See the lessons “Getting your part ready to 3D print” and “Arrange parts and export for printing”.
Other CAD programs	STEP, which keeps exact shapes. Ironcast also opens STEP files.
Games and renders	OBJ or GLB.
Drawings	PDF, DWG, DXF or SVG from the Drawing and 2D Drafting workspaces.
Machining	G-code from the Manufacture workspace.

Versions

To keep a copy of a design as it is right now, right-click the top item in the browser and choose **Create Version**. Ironcast saves a numbered copy beside your designs, with v1, v2 and so on added to its name.

More than one design at a time

Each open design has a tab at the top. To see two designs side by side, open the Data panel, right-click a design and choose **Open in new window**.

CHAPTER 1

First steps

Moving around, sketching and making your first solids. Start here if Ironcast is new to you.

Getting around: orbit, pan and zoom
Use your mouse, the ViewCube and the navigation bar to look at a model from any angle.

Your first sketch
Start a sketch on a plane, draw a rectangle and a circle, add a dimension, and finish the sketch.

Turn a sketch into a solid
Draw a rectangle, Extrude it into a solid with a distance, then Press Pull a face to change its size.

Rounding and bevelling edges

Start with a box, round some edges with Fillet and bevel others with Chamfer.

Adding holes

Use the Hole command to drill a hole in a face, set its diameter and depth, then add a second one.

Hollowing out a part

Use Shell to scoop out the inside of a solid block, open the top face, and choose how thick the walls are.

Undo, redo and the timeline

Build two quick features, undo and redo with Ctrl+Z and Ctrl+Y, then use the timeline to go back, edit an earlier feature and watch the model update.

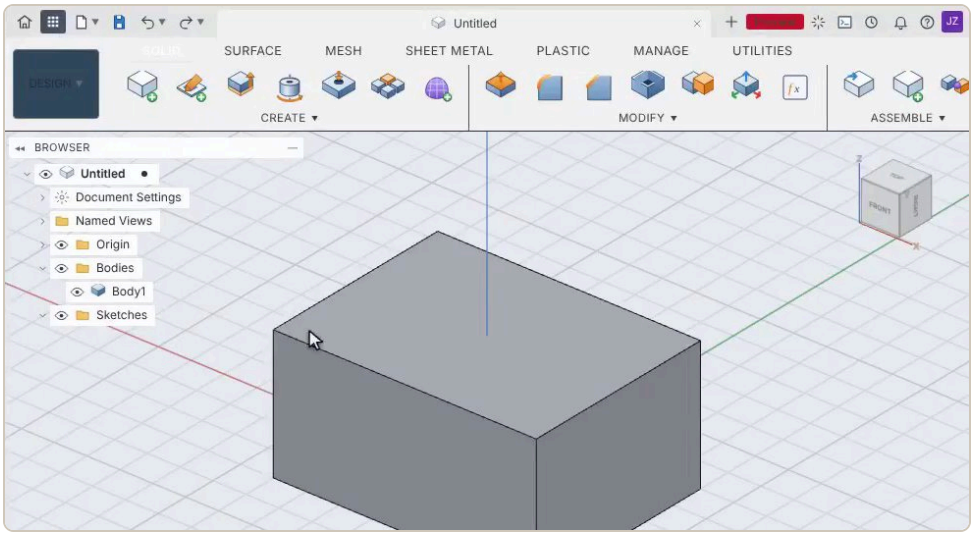
Selecting exactly what you want

Click to pick, use Shift and Ctrl to add and remove, drag window and crossing boxes, and use selection filters.

LESSON 1 Video in Ironcast: Home › Learn › “Getting around: orbit, pan and zoom” (1:06)

Getting around: orbit, pan and zoom

Use your mouse, the ViewCube and the navigation bar to look at a model from any angle.



Welcome to Ironcast. In this first lesson you'll learn how to move around your design.

1. Here's a simple block to practice on. The big area in the middle is the canvas, where your model lives.
2. To zoom, roll the mouse wheel. Ironcast zooms toward wherever your pointer is.
3. To pan, hold down the mouse wheel like a button, and drag. The model slides across the screen.
4. To orbit, and spin the model around, hold Shift, then hold the mouse wheel and drag.
5. Up in the top right corner is the ViewCube. Click any face of the cube to look straight at that side.
6. Click the little house next to the cube to jump back to the home view any time you get lost.

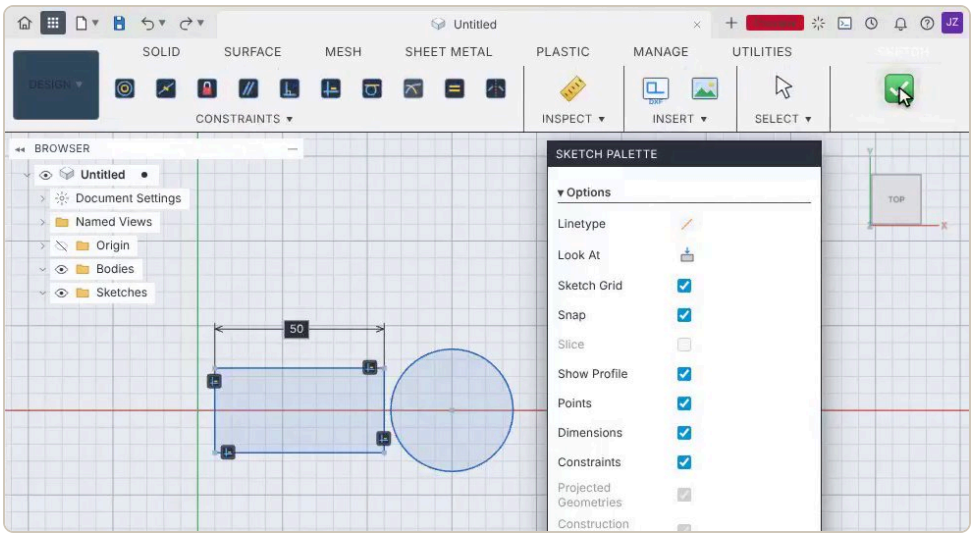
7. At the bottom is the navigation bar. Fit, or the F6 key, zooms so the whole model fills the screen.

That's it. Wheel to zoom, hold the wheel to pan, add Shift to orbit, and use the ViewCube to snap to a view. Next, you'll draw your first sketch.

LESSON 2 Video in Ironcast: Home › Learn › “Your first sketch” (1:20)

Your first sketch

Start a sketch on a plane, draw a rectangle and a circle, add a dimension, and finish the sketch.



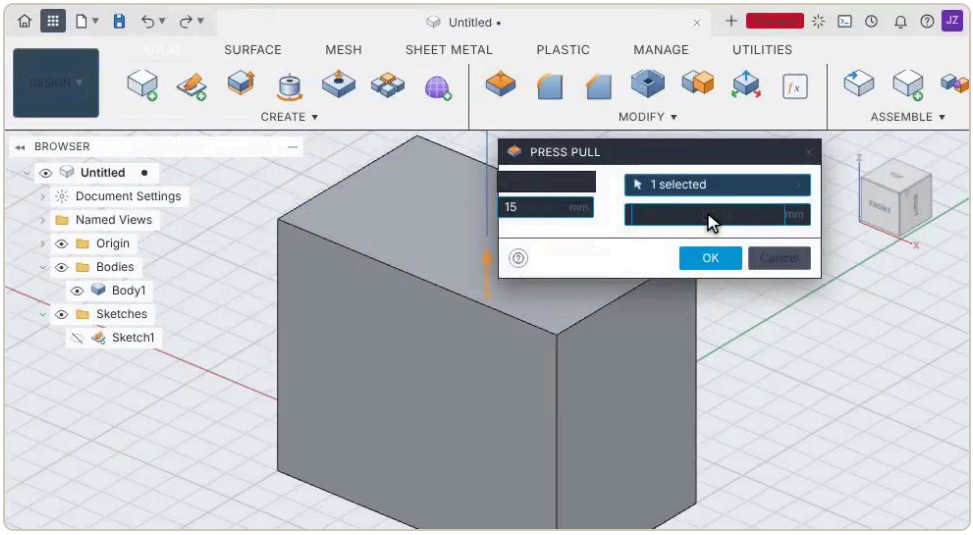
In this lesson you'll make your first sketch. Almost every part in Ironcast starts as a flat drawing like this.

1. Up in the toolbar, click Create Sketch.
2. Ironcast shows three flat planes and asks which one to draw on. Let's pick the blue one, which is the floor.
3. The view turns to face the plane, and a new Sketch tab appears. Hold the mouse wheel and drag to slide the sketch over, so there's room to draw.
4. Let's draw a rectangle. Click Rectangle, click once for one corner, then click again for the opposite corner.
5. Now a circle. Click Circle, click where the center goes, then click again to set the size.
6. Right now the shapes can be any size. Dimensions let you control that. Press D to start the dimension tool.
7. Click the top edge of the rectangle, then click above it to place the dimension.
8. A box appears where you can type the exact length. Type 50, and press Enter.
9. The rectangle stretches to exactly 50 mm wide.
10. Last, find Finish Sketch at the far right end of the toolbar, and click it to go back to the model.

That's your first sketch. Next, you'll turn it into a solid shape.

Turn a sketch into a solid

Draw a rectangle, Extrude it into a solid with a distance, then Press Pull a face to change its size.



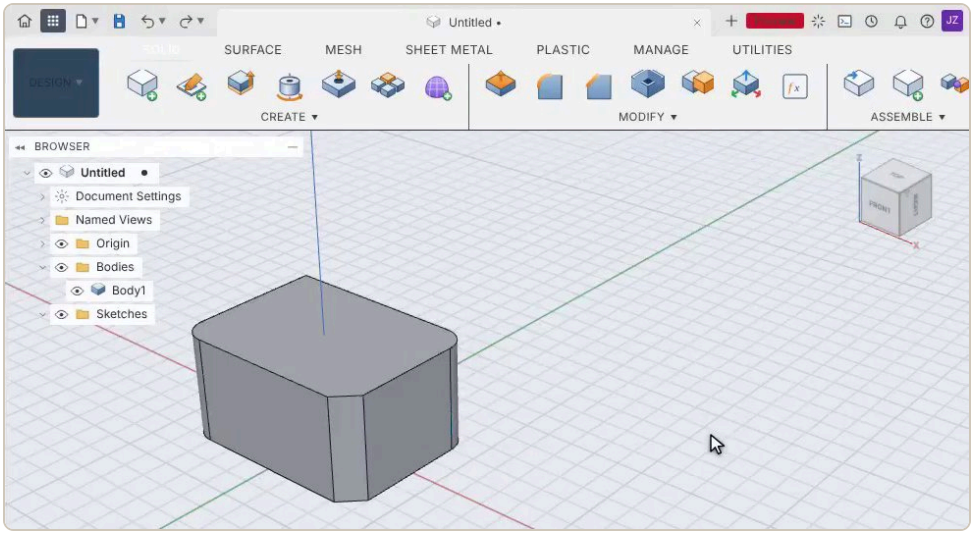
In this lesson you'll turn a flat sketch into a real 3D solid.

1. First, click Create Sketch. Then click the floor plane, the blue square, to draw on it.
2. Ironcast turns to face the sketch. Roll the mouse wheel to zoom in, then hold the wheel and drag to slide the grid over.
3. Choose the Rectangle tool and click once to place a corner. Then type 50 for the width, press Tab, type 30 for the height, and press Enter.
4. That closed shape is called a profile. Slide the ribbon over to find Finish Sketch, and click it.
5. Back on the Solid tab, click Extrude. You can also press the E key.
6. The Distance box says how tall to make it. Type 25, which means 25 mm.
7. Click OK, and your flat rectangle is now a solid block.
8. Click the little house by the ViewCube to look at it from an angle.
9. Now for Press Pull. It pushes or pulls a flat face. Click Press Pull, then click the top of the block.
10. Type 15 for the distance, and the top lifts up and makes the block taller.

That's the basic recipe. Sketch a profile, Extrude it to make a solid, then Press Pull a face to change it. Next, you'll learn to add rounded edges.

Rounding and bevelling edges

Start with a box, round some edges with Fillet and bevel others with Chamfer.



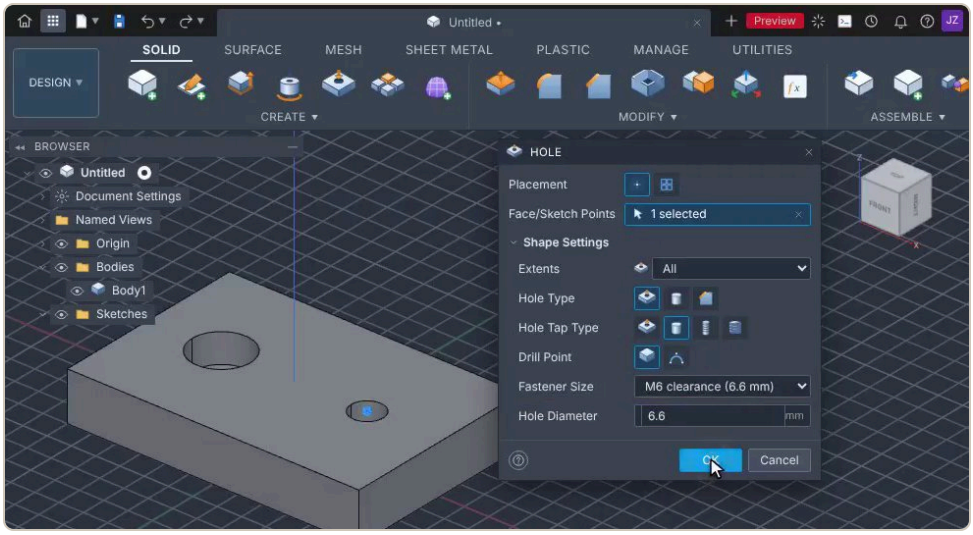
In this lesson you'll learn two ways to soften the sharp edges of a part, Fillet and Chamfer.

1. Here's a plain box. Every edge is sharp, which can be rough to hold and hard to make. Let's soften a few of them.
2. Fillet rounds an edge. Click Fillet in the Modify panel, or just press the F key.
3. Now click the edges you want rounded. I'll pick the tall edge on the right.
4. Click a second edge to add it. This one is the tall edge on the left.
5. Next, set the radius. That's how round the corner will be. Type 5 mm, and the preview updates right away.
6. Happy with it? Click OK. Those two corners are now smoothly rounded.
7. Now for Chamfer. It cuts a flat, angled bevel instead of a curve. You'll find it right beside Fillet.
8. Click the edge you want to bevel. I'll take the tall edge at the front.
9. Then set the distance, which is how far the bevel cuts into each face. Four millimetres will do.
10. Click OK. Now you can see the difference. The fillet is a smooth curve, and the chamfer is a flat angle.

That's it. Use Fillet for soft, rounded edges, and Chamfer for crisp, angled ones. Pick whichever suits your part.

Adding holes

Use the Hole command to drill a hole in a face, set its diameter and depth, then add a second one.

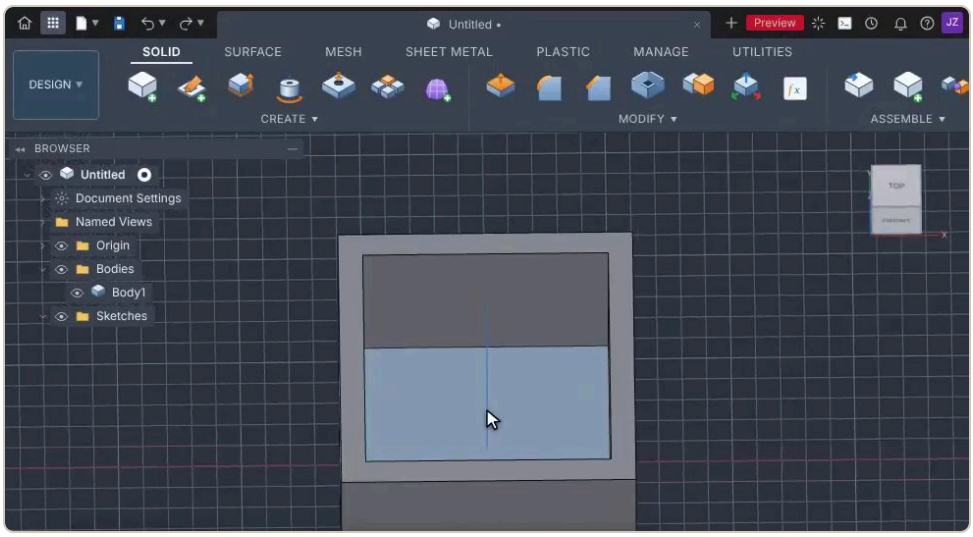


In this lesson you'll learn how to drill holes in your design.

1. Here's a flat plate. We'll put a hole in the top of it.
2. On the Solid tab, in the Create group, click Hole. You can also just press the H key.
3. A Hole panel opens. Now click on the top face of the plate, right where you want the hole to go.
4. A little circle shows up on the face. That's where your hole will be drilled.
5. Next, the size. Click in Hole Diameter and type 12 mm.
6. Now the depth. Change Extents from All to Distance. All makes a hole that goes right through.
7. A new Hole Depth box appears. Type six, so the hole stops partway into the plate.
8. Everything looks right, so click OK to finish the hole.
9. Let's add a second hole. Click Hole again, and click a spot on the other side of the plate.
10. This time, pick a standard screw size from the Fastener Size list. Ironcast fills in the diameter for you. Leave Extents on All, so this hole goes all the way through.
11. Click OK, and there are your two holes.
12. Now spin the view a little, and you can see the difference. One hole has a floor at the bottom, and the other goes all the way through. Next, you'll learn about fillets.

Hollowing out a part

Use Shell to scoop out the inside of a solid block, open the top face, and choose how thick the walls are.



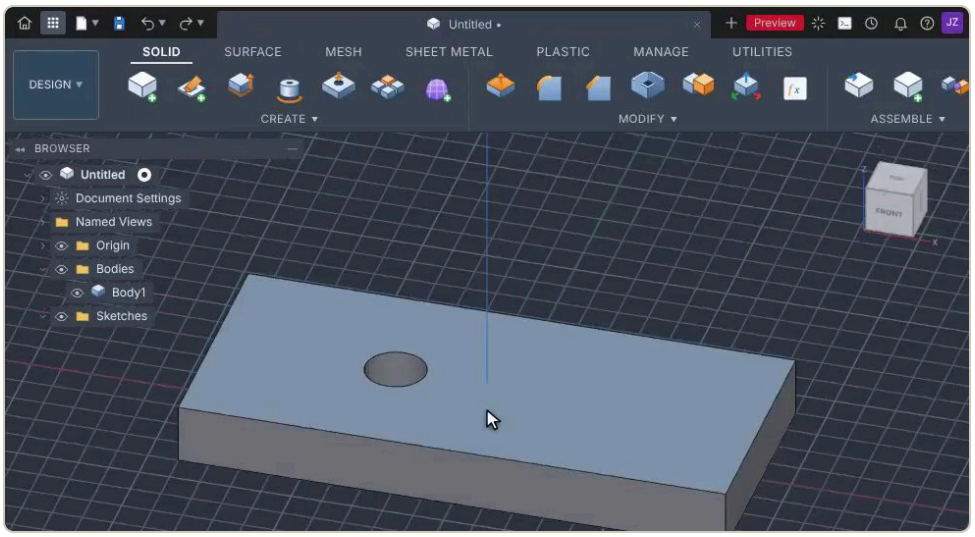
In this lesson you'll learn how to hollow out a solid shape, like making a box with an open top.

1. We start with a plain solid block. Right now it's solid all the way through.
2. On the Solid tab, in the Modify group, click Shell. It hollows a body and opens the faces you choose.
3. Now click the top face of the block. This is the face that will be removed, leaving an open top.
4. The Shell panel shows your choice, and the block is already hollow in the preview. The walls start out 2 mm thick.
5. Let's make the walls thicker so they're easy to see. Click in Inside Thickness and type 5 mm.
6. The Direction is set to Inside, so the outside of the block keeps its size, and the walls grow inward.
7. That looks right, so click OK to finish the shell.
8. Now let's look inside. Hold Shift, hold the mouse wheel, and drag up to tip the top of the model toward you.
9. There's the hollow inside, with walls and a floor, all 5 mm thick. Orbit a little more to see it from another side.

That's Shell. Pick the faces to open, set the thickness, and click OK. Next, you'll round off sharp edges with fillets.

Undo, redo and the timeline

Build two quick features, undo and redo with Ctrl+Z and Ctrl+Y, then use the timeline to go back, edit an earlier feature and watch the model update.



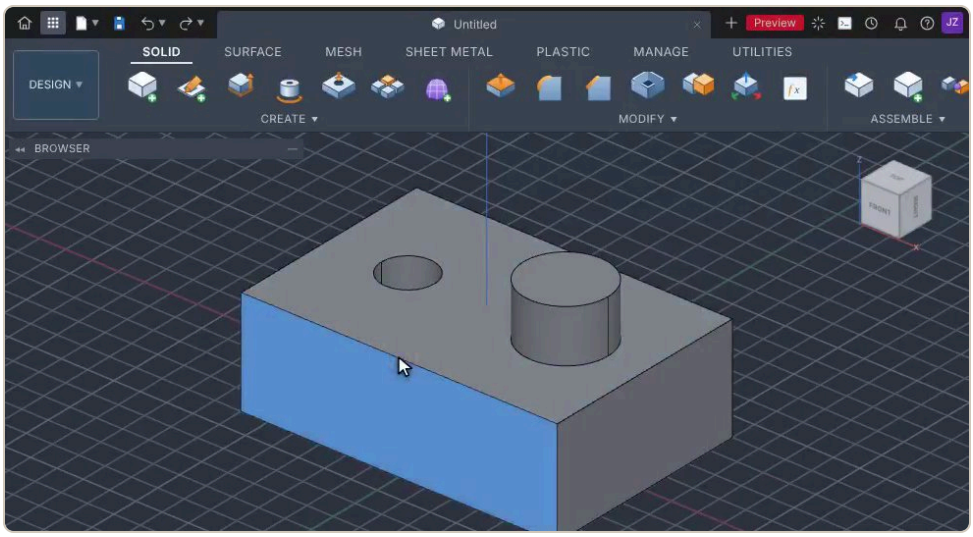
In this lesson you'll learn how to undo your changes, and how to travel back through your design's history.

1. First, let's build two quick features. Open the Create menu and choose Box.
2. Type 60 for the length, 40 for the width, and ten for the height. Then click OK.
3. That's feature number one, a flat block. Feature number two will be a hole. Press H, then click the top of the block.
4. Type ten for the diameter, and click OK.
5. Now look at the bar along the bottom. This is the timeline. Every feature you make adds an icon to it, so here is the box, and next to it, the hole.
6. Made a mistake? Press Control Z to undo. The hole disappears, and its icon leaves the timeline.
7. Changed your mind? Press Control Y to redo, and the hole comes right back.
8. The timeline can do more than undo. Click Step Back, the little arrow on the left of the bar.
9. The marker moved back one step. The hole icon is greyed out, and the model looks the way it did before the hole. Click Step Forward to return to the end.
10. Here's the best part. You can change an earlier feature. Double-click the first icon, the box, to open it again.
11. Change the length from 60 to 90, and click OK.
12. The block grew longer, and the hole is still in place. Ironcast rebuilt every later feature for you.

That's it. Control Z and Control Y undo and redo, and the timeline lets you step back, and edit any earlier feature. Next, you'll learn how to save your work.

Selecting exactly what you want

Click to pick, use Shift and Ctrl to add and remove, drag window and crossing boxes, and use selection filters.



In this lesson you'll learn how to select exactly the faces, edges and bodies you want.

1. Here's a block with a round boss on top and a hole through it. We'll use it to practice picking things.
2. To select something, just click it. Click the front face of the block, and it turns blue.
3. Click somewhere else, and the selection moves. A plain click always picks just one thing.
4. To add more, hold Shift and click. Let's add the right face, and the top of the round boss.
5. To take something out, hold Ctrl and click it again. Shift works the same way, so use whichever you like.
6. Click empty space to clear everything. Edges can be picked too. Click right on the rim of the hole, and that whole circle is selected.
7. To pick many things at once, drag a box. Drag from left to right for a window selection. It only picks edges that are completely inside the box.
8. Now drag the same box from right to left. This is a crossing selection. The box turns green, and it picks every edge it touches, even partly.
9. Box selection also reaches through the part. Drag a window around everything, and even the hidden edges at the back get selected.
10. Selection filters control what a click can pick. First, clear the selection, then click on this edge. The edge itself is chosen.
11. Now open the Select menu at the right of the toolbar, then Selection Filters, and untick Body Edges.
12. Click the very same spot again. With edges switched off, Ironcast picks the face underneath instead.
13. Open the filters again and tick Body Edges to turn edges back on. That's selecting: click to pick, Shift or Ctrl to add and remove, drag boxes for window and crossing, and filters to limit what you can pick.

CHAPTER 2

Sketching in depth

Rules that hold a sketch together, smooth curves, and numbers you can change later.

Sketch dimensions and constraints
Lock a rough sketch in place with Coincident, Horizontal, Equal and Tangent constraints plus dimensions, until the sketch is fully constrained.

Sketch: smooth joins with Curvature
Draw a line and a spline that starts at its end, make the join perfectly smooth with the Curvature constraint, compare it with Tangent on an arc, then extrude the shape.

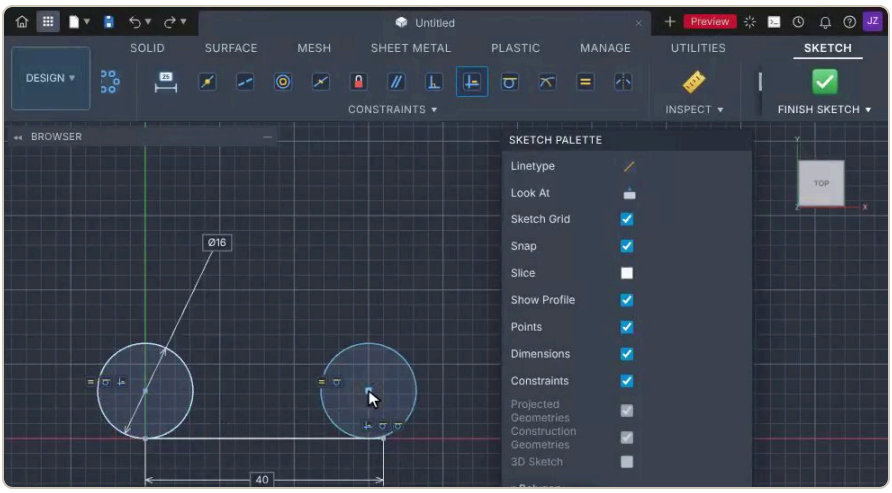
Parameters
Add user parameters in Modify > Change Parameters, type their names and expressions like width/2 into Box and Shell, then change one value and watch the model update.

User Coordinate System
Make your own origin on a face with Construct > User Coordinate System, turn it 30 degrees, then sketch on its XY plane and extrude a boss that comes out at the angle.

LESSON 9 — Video in Ironcast: Home › Learn › “Sketch dimensions and constraints” (1:34)

Sketch dimensions and constraints

Lock a rough sketch in place with Coincident, Horizontal, Equal and Tangent constraints plus dimensions, until the sketch is fully constrained.



In this lesson you'll lock a sketch in place with dimensions and constraints.

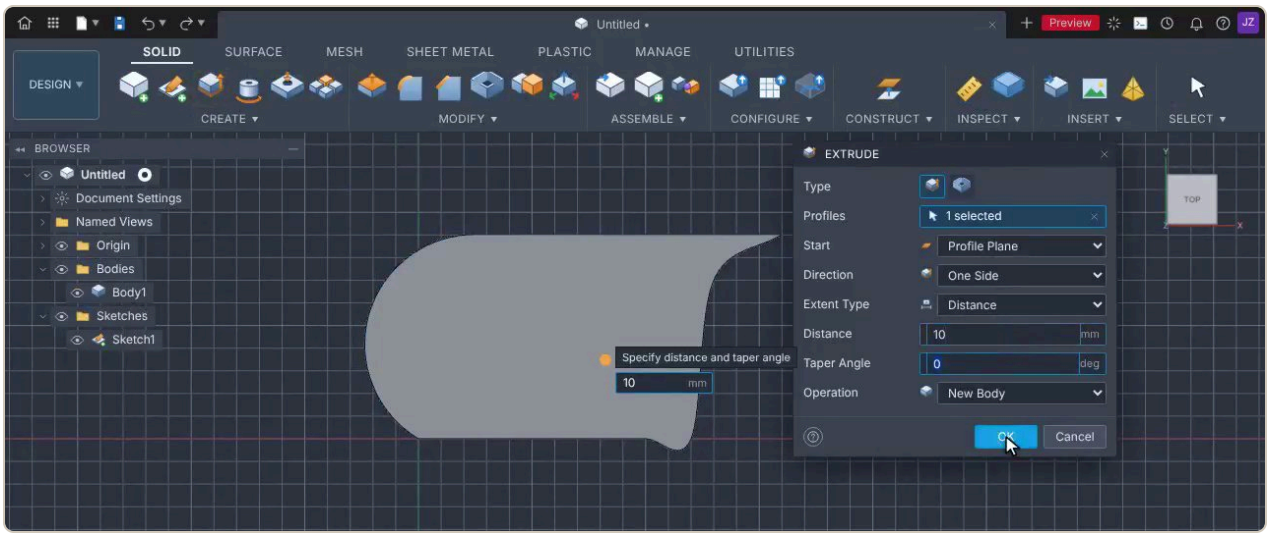
1. Here's a rough sketch: a tilted line and two circles. Blue means a shape is still free, so you can drag it around.

2. The sketch palette agrees. It says three curves are free to move.
3. Constraints are rules for your shapes. Click Coincident, then click the end of the line, and then the origin point. The end snaps onto the origin.
4. Next, click Horizontal Vertical, and click the line. It levels out.
5. Now a dimension. Click Sketch Dimension, click the line, then click below it to place the number.
6. Type 40 and press Enter. The line turns white. That means it's locked.
7. On to the circles. Click Equal, then click one circle and the other. Now they're the same size.
8. Give one circle a size. With Sketch Dimension, click the circle and place the number.
9. Type 16 and press Enter. Both circles change, because they're equal.
10. Click Tangent, then click the line and a circle. The circle drops down to touch the line. Then do the same for the other circle.
11. Last, Horizontal Vertical also works on two points. Click a circle's center, then the line end below it, and they line up. Do the same on the right.
12. Everything is white now, and the palette says Fully constrained. Your sketch is exact, and it can't change by accident.

LESSON 10 Video in Ironcast: Home › Learn › “Sketch: smooth joins with Curvature” (1:33)

Sketch: smooth joins with Curvature

Draw a line and a spline that starts at its end, make the join perfectly smooth with the Curvature constraint, compare it with Tangent on an arc, then extrude the shape.



In this lesson you'll make curves flow into each other smoothly, with the Curvature constraint.

1. We're in a sketch on the floor plane. Click the Line tool, click the origin, then click 40 mm to the right, and press Escape.
2. Now open Create, Spline, and choose Fit Point Spline. Start at the end of the line, click four more points up to the right, and press Enter.

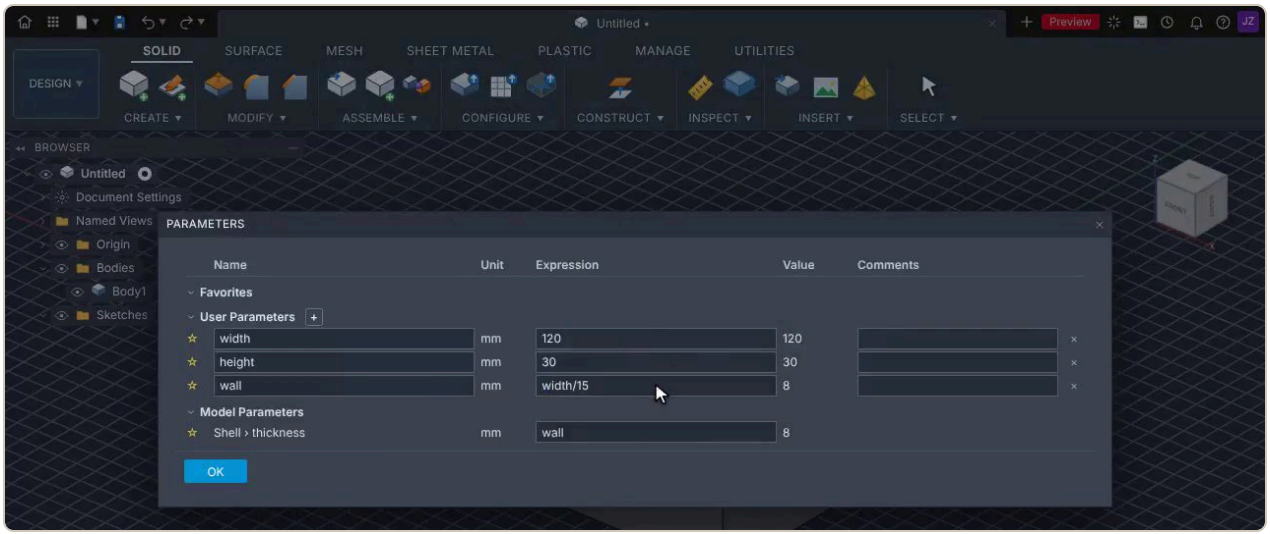
3. Look at the join. The line runs flat, but the spline leaves it at an angle, so there's a sharp corner.
4. Open Constraints and choose Curvature. Click the spline first, then the line.
5. Now the spline leaves the line in the same direction and with the same bend, so the join is smooth.
6. For comparison, draw a line back across the top, from the end of the spline to above the origin.
7. Pick the 3-Point Arc. Click the end of the top line, then the origin, then a point out to the left.
8. There's a corner where the arc meets the top line. Open Constraints, choose Tangent, and click the top line, then the arc.
9. Now the arc meets the line in the same direction. Tangent only matches direction, while Curvature also matches how sharply the curve bends.
10. Click Finish Sketch. Then click Extrude, type ten for the distance, and click OK.

That's it. Use Curvature where a spline should flow out of a line or an arc with no visible seam.

LESSON 11 Video in Ironcast: Home › Learn › “Parameters” (1:06)

Parameters

Add user parameters in Modify > Change Parameters, type their names and expressions like width/2 into Box and Shell, then change one value and watch the model update.



Let's drive a model with named parameters.

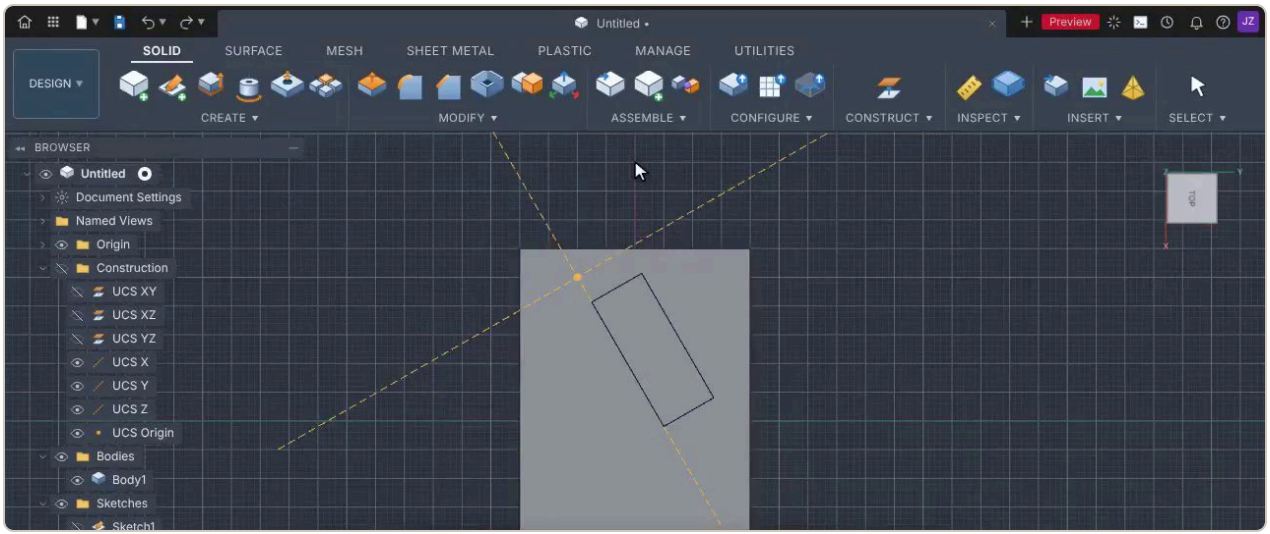
1. Open Modify, choose Change Parameters, and click plus three times.
2. Name them width, height and wall. Set them to 80, 30 and three.
3. Click OK. Then open Create, and choose Box.
4. Type width for the length, width slash two for the width, and height for the height.
5. Click OK. Open Modify, choose Shell, and click the top face.
6. Type wall for the thickness, and click OK.
7. Back in Change Parameters, the shell thickness now reads wall.

8. Change width to 120, and click OK.
9. The box grows, and its width follows at half, 60 mm.
10. Now change wall to six. Every wall thickens at once.
11. Expressions work here too. Set wall to width divided by 15, and the walls grow to eight.
12. Name your key sizes once, and change the whole design from one place.

LESSON 12 Video in Ironcast: Home › Learn › “User Coordinate System” (1:52)

User Coordinate System

Make your own origin on a face with Construct > User Coordinate System, turn it 30 degrees, then sketch on its XY plane and extrude a boss that comes out at the angle.



In this lesson you'll make your own coordinate system, and use it to build a feature at an angle.

1. Here's a plain block. Its edges line up with the main X and Y axes. We want to add a boss turned 30°.
2. Open the Construct menu, and choose User Coordinate System.
3. For the origin, click the top of the block, near the front left corner. The new Z axis follows the face, so it points straight up.
4. Set Rotate about Z to 30°. The new X and Y axes swing around.
5. The offsets slide the origin. Type minus five for Offset X, then click OK.
6. Look in the browser, in the Construction folder. The UCS has its own XY, XZ and YZ planes, its own three axes, and an origin point.
7. Now click Create Sketch, and pick the UCS XY plane. Click the part of the orange square that hangs past the block.
8. The view turns to look straight down the new Z axis, so the block looks turned. The sketch grid lines up with the UCS.
9. Choose Rectangle, and click just right of the origin. Type 25, press Tab, type ten, and press Enter.
10. Slide the ribbon over, and click Finish Sketch.

11. Click Extrude. Type 15 for the distance, and click OK.
12. Click the home button by the ViewCube, and hide the Construction folder with its eye icon.
13. Now look from the top. The new boss sits at 30° to the block, lined up with the UCS instead of the main axes.

That's a User Coordinate System. Put an origin anywhere, turn it, and sketch or measure from its planes.

CHAPTER 3

More ways to shape a solid

Turned parts, swept and lofted shapes, patterns, text and threads, and editing faces directly.

Revolve a profile

Sketch half of a knob next to an axis, then use Revolve to spin it into a round solid.

Sweep and loft

Sweep a circle along a bent path to make a tube, then loft from a square to a circle to make a smooth transition.

Mirror and pattern

Copy a pin into a grid with Rectangular Pattern and cut it into holes, spin a peg around a disc with Circular Pattern, then Mirror a whole plate.

Coil, Text and Thread

Make a spring with Create > Coil, write sketch text on a face and raise it with Emboss, and cut a modeled M12 thread on a shaft with Create > Thread.

Replace Face: reshape a face to match another

Make a block and a flat plane, then use Replace Face so the block's top face moves up to meet the plane.

Boundary Fill: solid from enclosed space

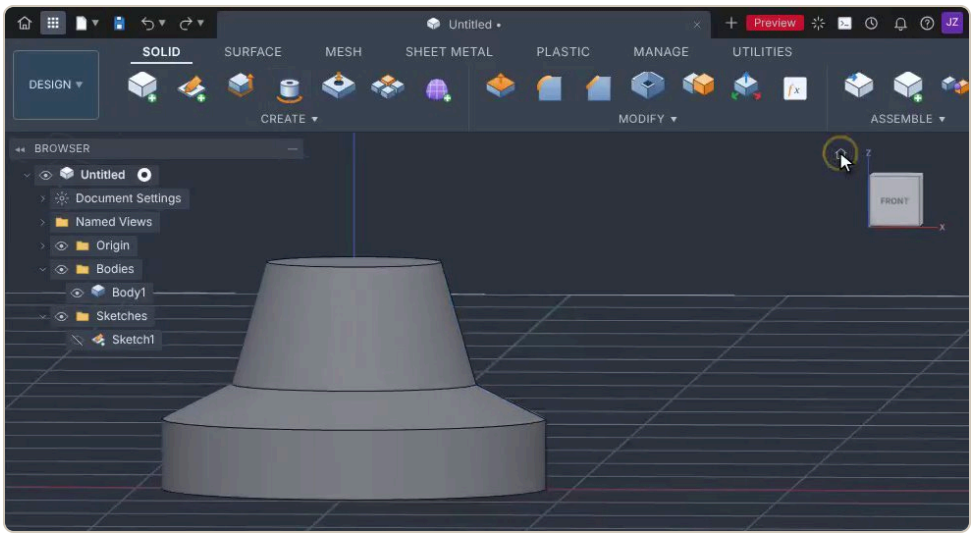
Turn an empty room inside a tray and lid into a solid body of its own with Boundary Fill.

Simplify parts with Remove Features

Fill small holes with Modify > Simplify > Remove Features, swap a part for a box or cylinder with Replace with Primitives, and edit both from the timeline.

Revolve a profile

Sketch half of a knob next to an axis, then use Revolve to spin it into a round solid.

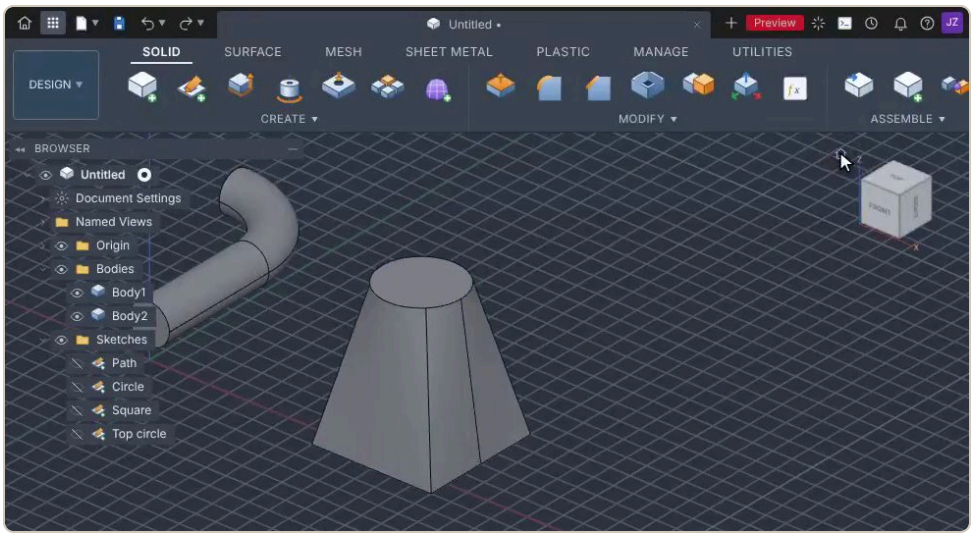


In this lesson you'll make a round part with Revolve. You draw just half of the shape, and Ironcast spins it around a line.

1. Click Create Sketch, then click the green upright plane. That's the front plane.
2. Roll the mouse wheel to zoom in, then hold the wheel and drag to move the center point down to the left.
3. See the green line going straight up? That's the line we'll spin around. We'll draw the right half of a knob, right up against it.
4. Pick the Line tool and click from point to point. Start on the green line, go out along the bottom, up the side, and back in to the green line.
5. Click the first point again to close the shape. It fills in with color, which means it's a closed profile.
6. Slide the toolbar over and click Finish Sketch.
7. Now, on the Solid tab, click Revolve.
8. The Revolve panel already picked our profile, so Profile says one selected.
9. Sketch Axis is set to the sketch Y axis. That's the green line we drew against, so the shape spins around it.
10. Type is set to Full, which spins it all the way around. Choose Angle instead if you only want part of a turn.
11. Click OK, then click the little house by the view cube to see it from an angle.
12. There's your knob. One flat sketch turned into a fully round solid. Revolve works great for knobs, wheels, bottles and vases.

Sweep and loft

Sweep a circle along a bent path to make a tube, then loft from a square to a circle to make a smooth transition.

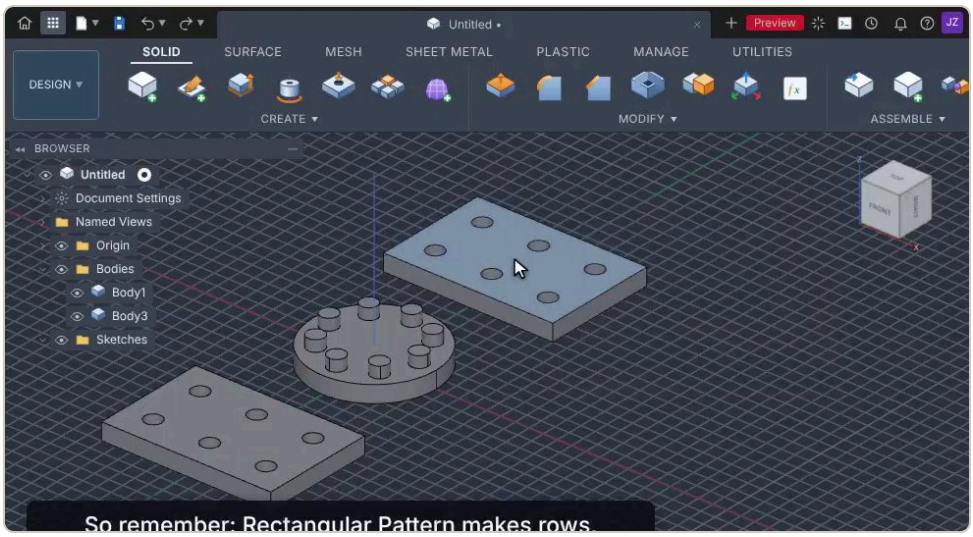


In this lesson you'll learn two handy tools. Sweep moves a shape along a path, and Loft blends from one shape into another.

1. I've drawn two sketches ahead of time. A small circle standing up, and a bent path on the floor that starts right at the circle.
2. Open the Create menu and choose Sweep.
3. First, click inside the circle. That's the profile, the shape that gets moved along.
4. Next, click the Path box, then click the bent line on the floor.
5. Ironcast shows a preview of a tube that follows the path all the way around the bend. Click OK.
6. Now for Loft. Over here there's a square on the floor, and a circle floating above it on a higher plane.
7. Open the Create menu again, and this time choose Loft.
8. Click the square first, then click the circle. The panel lists them in that order, from bottom to top.
9. The preview blends smoothly from the square at the bottom into the round top. Click OK.
10. Click the little house to see both parts together.
11. Use Sweep for things like pipes, handles and wires. Use Loft for funnels, bottles, and anything that changes shape from one end to the other.

Mirror and pattern

Copy a pin into a grid with Rectangular Pattern and cut it into holes, spin a peg around a disc with Circular Pattern, then Mirror a whole plate.



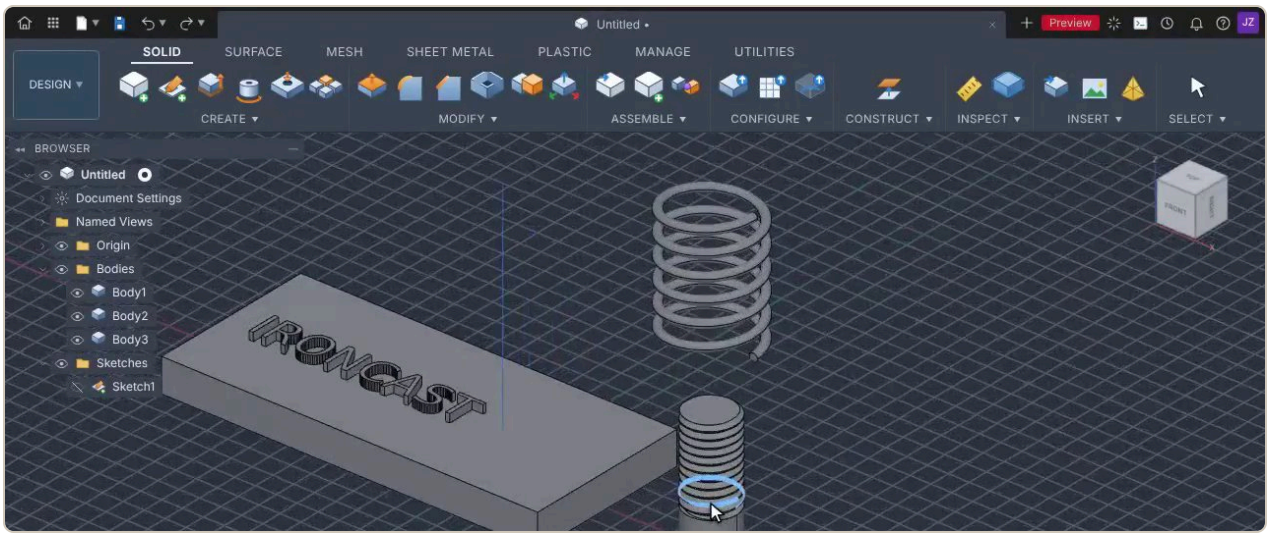
In this lesson you'll learn to copy things the smart way, with patterns and mirror.

1. Here's a flat plate with one small pin standing on it, and a round disc with one peg.
2. Let's copy the pin into rows. On the Solid tab, click Rectangular Pattern, then click the pin.
3. Right away you see three pins in a row. The first Quantity and Spacing control the X direction: three pins, 20 mm apart.
4. The second pair controls the Y direction. Change that Quantity to two, and now there are six pins. Click OK.
5. To turn the pins into holes, open the Modify menu and choose Combine.
6. Click the plate for the Target Body, then click a pin for the Tool Bodies.
7. Set Operation to Cut, and click OK. The pins are cut out of the plate, leaving six neat holes.
8. Next, the disc. Open the Create menu, go to Pattern, and choose Circular Pattern. Then click the disc.
9. It spins copies around the Z axis, the line through the middle. Quantity starts at six. Change it to eight, and click OK.
10. Last, Mirror. Open the Create menu and choose Mirror, then click the plate.
11. Change Mirror Plane to the X Z plane, which stands right through the middle of the disc. Click OK.
12. A flipped copy of the plate, holes and all, appears on the other side.

So remember: Rectangular Pattern makes rows, Circular Pattern goes around a center, and Mirror makes a flipped copy. They save you from drawing the same thing twice.

Coil, Text and Thread

Make a spring with Create > Coil, write sketch text on a face and raise it with Emboss, and cut a modeled M12 thread on a shaft with Create > Thread.

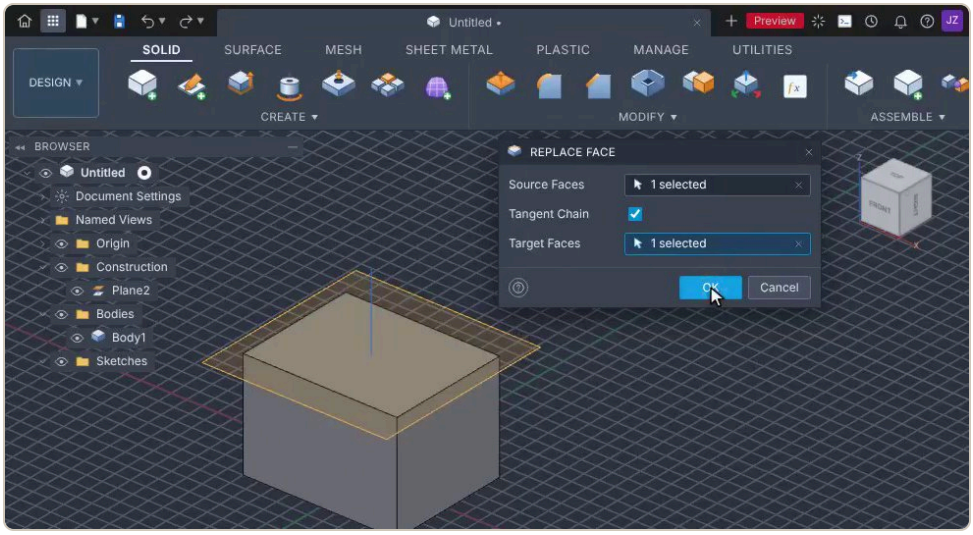


Let's make a spring, raised text, and a screw thread.

1. Here's a plate and a shaft. Open Create, and choose Coil, for a spring shaped helix.
2. Set Center Y to 60, to put it behind the plate, and click OK.
3. Now text. Click Create Sketch, then the top of the plate.
4. Open Create, choose Text, and click where the text starts.
5. Type IRONCAST. The letters preview on the plate.
6. Click OK, and then Finish Sketch.
7. Open Create and choose Emboss. All eight letters are picked for you.
8. Set Depth to two, and click OK. The letters stand up from the plate.
9. Last, a thread. Open Create, choose Thread, and click the side of the shaft.
10. Keep Modeled on, pick size M12 to match the shaft, and set Length to 20.
11. Click OK, and a real thread is cut into the shaft.
12. Coil, Emboss and Thread all stay in the timeline, so you can edit them any time.

Replace Face: reshape a face to match another

Make a block and a flat plane, then use Replace Face so the block's top face moves up to meet the plane.

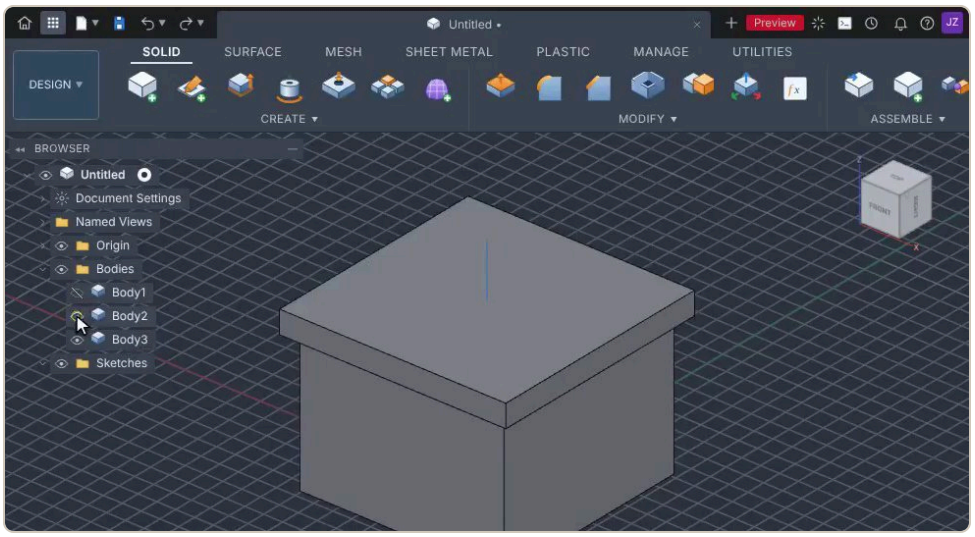


In this lesson you'll learn how to reshape a face so it matches another one.

1. Here's a simple block. Say we want its top face to be higher, and to line up with a flat surface.
2. First we need that surface. Open the Construct menu and choose Offset Plane.
3. The plane starts on the base plane. Type 35 for the distance, so it floats above the block, then click OK.
4. Now there's a plane hovering above the block. Next, open the Modify menu and choose Replace Face.
5. The Replace Face panel is asking for two things. Source Faces is the face that will move. Click the top of the block.
6. Now click in Target Faces, then click the plane. This is the surface the block's face should follow.
7. Click OK.
8. The top of the block stretched up and now sits exactly on the plane. Spin the view to see that the sides grew with it. Next time, you'll try more new tools.

Boundary Fill: solid from enclosed space

Turn an empty room inside a tray and lid into a solid body of its own with Boundary Fill.

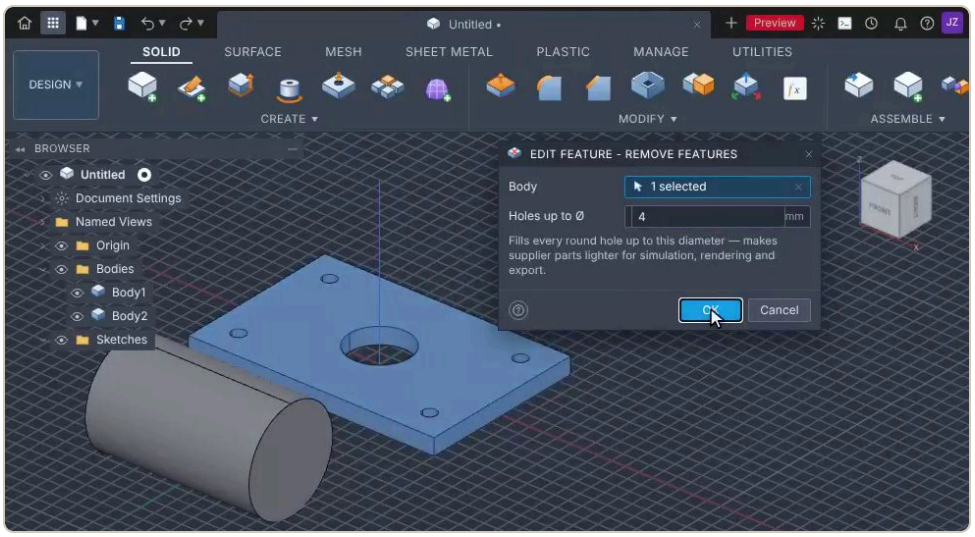


In this lesson you'll turn an empty space inside your design into a solid body, using Boundary Fill.

1. Here's an open tray, like a box with no lid. Let's close it up with a lid.
2. On the Solid tab, click Box. Set Center Z to 40, the size to 60 × 60 by six, and the operation to New Body.
3. Click OK. Now the tray and its lid are two separate bodies that wall in a hidden room.
4. To find the Boundary Fill command, open the Create menu on the Solid tab, and choose Boundary Fill.
5. Tool Bodies is asking for the bodies that wall in the space. Click the tray first.
6. Then click the lid.
7. Ironcast now lists the closed regions these two bodies make, called cells, with the volume of each one.
8. The big empty room inside is the one we want. Tick the cell with the largest volume.
9. Turn off Remove Tools, so the tray and lid stay put. Leave the operation on New Body.
10. Click OK.
11. Nothing seems to have changed, because the new body is hiding inside. In the Browser, click the eye next to the tray.
12. Now click the eye next to the lid.
13. And there it is. A solid block, the exact shape of the empty room. Next time you need a plug, mould or filler, you can make it this way.

Simplify parts with Remove Features

Fill small holes with Modify > Simplify > Remove Features, swap a part for a box or cylinder with Replace with Primitives, and edit both from the timeline.



In this lesson you'll simplify parts, filling small holes and swapping detailed shapes for simple blocks.

1. This plate has six small holes and one big hole in the middle. Next to it is a shaft with a collar.
2. For simulation or a light export, small details just slow things down. Open Modify, then Simplify, and choose Remove Features.
3. Click the plate. Every round hole up to 10 mm fills in right away.
4. The big hole is 22 mm across, so it stays open. Set Holes up to six, and click OK.
5. Now the shaft. Open Modify, Simplify, and choose Replace with Primitives.
6. Click the shaft. With Primitive set to Box, it turns into a simple block that encloses it. Click OK.
7. Both steps are features in the timeline. Double-click the last one to edit it.
8. Change Primitive to Cylinder. Now a round shape wraps the shaft. Click OK.
9. Remove Features can be edited the same way. Double-click it, and set Holes up to four.
10. The 5 mm corner holes come back, and only the 3 mm holes stay filled. Click OK.
11. Simplify never loses your original design. These steps live in the timeline, so you can change or delete them any time.

CHAPTER 4

Projects for 3D printing

Three small parts made start to finish, then laid out and exported for your slicer.

Project: a printable L-bracket
Put it all together: sketch an L profile, extrude it, add two screw holes and fillet the inside corner.

Project: a box with a lid
Make a box with Shell, then build a separate lid body that fits inside it with a small clearance.

A 3D-printed box with a snap fit
Hollow out a block with Shell, add a Lip on the rim and a Snap Fit hook, then check the walls with Thickness Analysis.

Arrange parts and export for printing
Lay several bodies out in neat rows with Modify > Arrange, see the Arrange group in the timeline, then save them all for your slicer with 3D Print.

Getting your part ready to 3D print
Export your part as an STL or 3MF file, understand units, and know which file to open in a slicer like Bambu Studio.

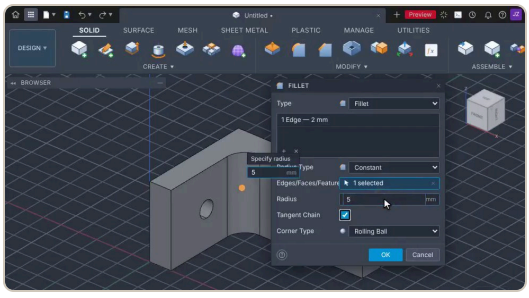
Bring in a 3D-print file
Drop a 3MF print file into Ironcast as a mesh body, look at the Mesh tools, stretch it with Non Uniform Scale, check its walls with Thickness Analysis and split it into Face Groups.

LESSON 20

Video in Ironcast: Home › Learn › “Project: a printable L-bracket” (1:49)

Project: a printable L-bracket

Put it all together: sketch an L profile, extrude it, add two screw holes and fillet the inside corner.



In this project you'll design a small L-shaped bracket that you could print on a 3D printer.

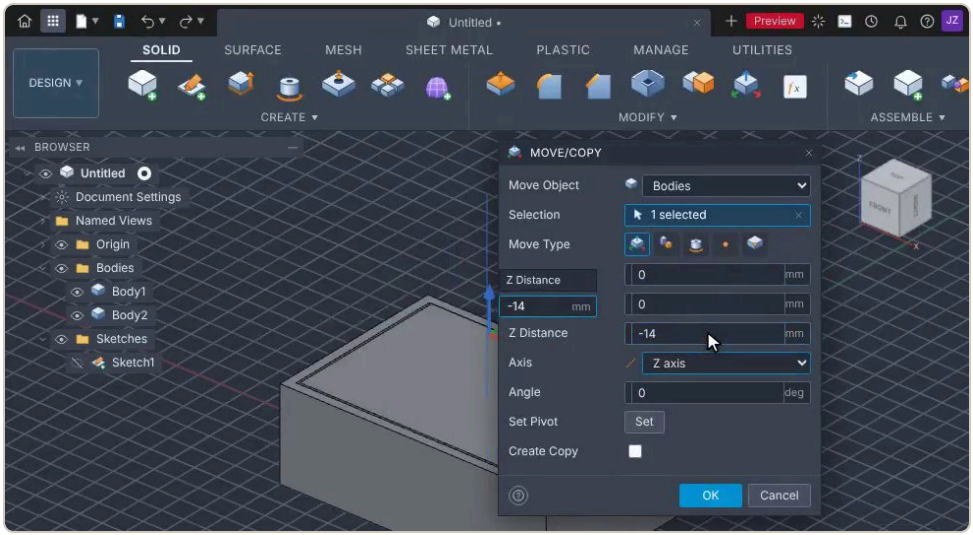
1. We'll sketch an L, extrude it, add two screw holes, and round the inside corner. Click Create Sketch, then pick the floor plane.

2. Roll the wheel to zoom in, then hold the wheel and drag to slide the grid into place.
3. The Line tool is already on. Click the corners of an L, one after another. The points snap to the grid, so the sizes come out neat.
4. Click the very first point again to close the shape. Ironcast fills it in. That's an L 40 mm long and 30 tall, with legs 6 mm thick.
5. Slide the ribbon over and click Finish Sketch.
6. Back on the Solid tab, click Extrude. Your L is already picked. Type 20 mm for the width, and click OK.
7. Click the little house by the ViewCube for an angled look. Then hold the wheel and drag to slide the bracket to the left, out from under the panels.
8. Now the screw holes. Click Hole, then click the inside face of the short leg. In Fastener Size choose M4, and click OK.
9. Do the same on the long leg. Click Hole, click its inside face, choose M4 again, and click OK.
10. Last, the fillet. A sharp inside corner is a weak spot. Click the house again to bring the bracket back to the middle, then click Fillet, and click the inside corner edge.
11. Type 5 mm for the radius, and click OK.
12. Spin the view to admire it. A bracket with two screw holes and a strong, rounded corner, ready to export and print.

LESSON 21 Video in Ironcast: Home › Learn › “Project: a box with a lid” (1:41)

Project: a box with a lid

Make a box with Shell, then build a separate lid body that fits inside it with a small clearance.



In this project you'll make a small box, and then a separate lid that fits it.

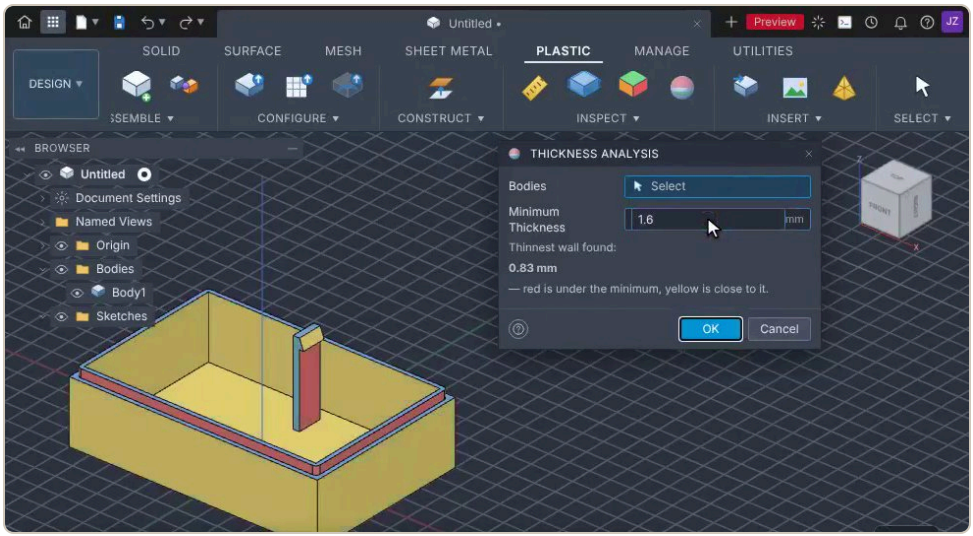
1. We start with a solid block, 60 mm long, 40 wide, and 20 tall.
2. First, hollow it out. Click Shell, then click the top face.
3. The walls are 2 mm thick, so the opening will be 56 × 36. Click OK.

4. Now for the lid. Click Create Sketch, then click the floor plane, the blue square, to draw on it.
5. The lid must be a little smaller than that opening. The small gap is called clearance. We'll leave 0.2 mm on each side.
6. Choose Rectangle and click a corner. Then type 55.6 for the width, press Tab, 35.6 for the height, and press Enter.
7. Slide the ribbon over and click Finish Sketch.
8. Click Extrude. The lid must be its own body, so change Operation to New Body. Otherwise it would join the box.
9. Set Start to Offset and type 30, so the lid starts in the air above the box. Make it four thick, and click OK.
10. The lid floats above the box as its own body. Now let's drop it in. Click Move, then click the lid.
11. Type −14 in Z Distance, and click OK.
12. The lid sits right in the opening, level with the rim, with a small gap all the way around. Orbit a bit and you can see it. That's a box with a lid that fits.

LESSON 22 Video in Ironcast: Home › Learn › “A 3D-printed box with a snap fit” (1:50)

A 3D-printed box with a snap fit

Hollow out a block with Shell, add a Lip on the rim and a Snap Fit hook, then check the walls with Thickness Analysis.



In this lesson you'll turn a solid block into a box that's ready to 3D print, with a lip and a snap fit hook.

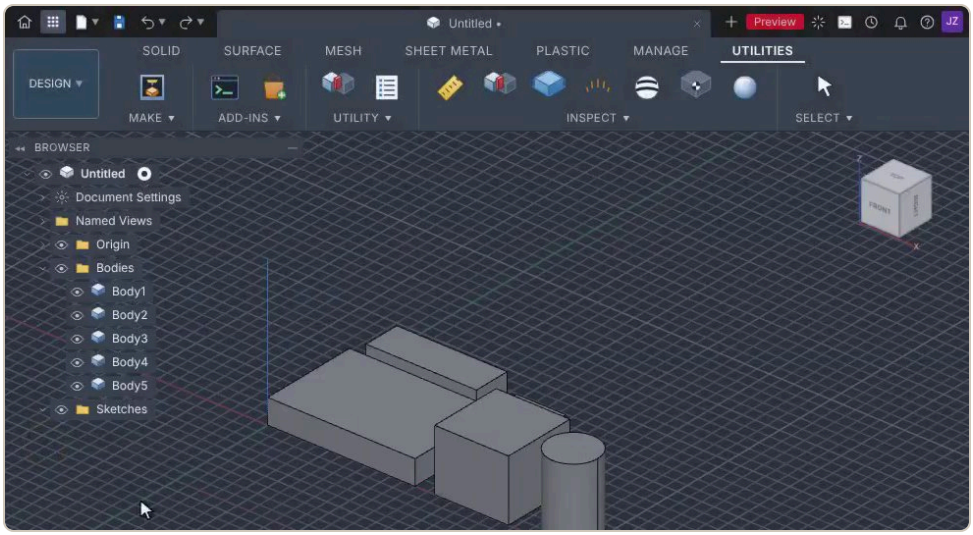
1. We're on the Plastic tab, which collects the tools for printed and molded parts. Here's a solid block to start from.
2. First, hollow it out. Click Shell, then click the top face. That face is removed and the block becomes an open box.
3. Inside Thickness is 2 mm, a good wall for printing. Click OK.

4. Next, a lip, so a lid can sit on the box and stay lined up. Open the Create menu and choose Lip.
5. Click the flat rim on top of the wall. A thin raised lip runs all the way around, along the inside edge.
6. The lip is 1 mm wide and two high. The lid would get a matching groove. Click OK.
7. Now a snap fit hook, to clip the lid shut. Open Create again and choose Snap Fit.
8. Click the floor of the box, near the back wall. A springy beam with a hook on top stands up from that spot.
9. Make it taller, so the hook reaches above the rim. Set Beam Length to 20, and click OK.
10. Before printing, check for walls that are too thin. Open the Inspect menu and choose Thickness Analysis.
11. The colors show how thick each part is. Green is fine, yellow is close to the minimum, and red is too thin. The 1 mm lip shows up in red.
12. Raise Minimum Thickness to 1.6, and the thin snap fit beam turns red too. Now you know which parts to make thicker.
13. Click OK to finish. That's a printable box with a lip and a snap fit, checked and ready to go.

LESSON 23 Video in Ironcast: Home › Learn › “Arrange parts and export for printing” (1:28)

Arrange parts and export for printing

Lay several bodies out in neat rows with Modify > Arrange, see the Arrange group in the timeline, then save them all for your slicer with 3D Print.



In this lesson you'll pack several parts neatly onto a print bed, then save them all in one file for your slicer.

1. Here are five parts of different sizes, scattered all over the place.
2. Open the Modify menu and choose Arrange.
3. Leave Bodies empty, and Arrange uses every visible body.
4. Set Envelope Width to 150 mm. That's how wide each row is allowed to get.

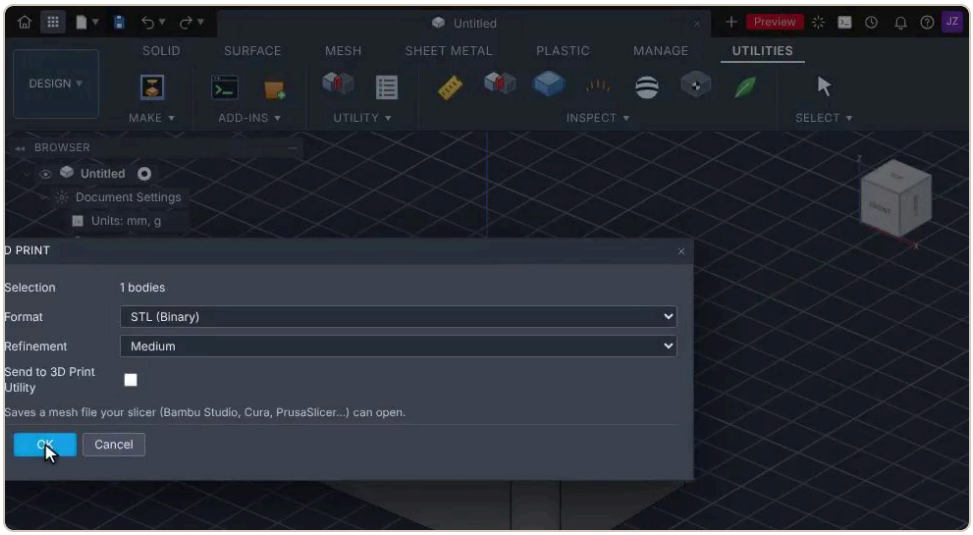
5. Set Spacing to eight, for a gap between the parts. Drop to Grid sets every part flat on the grid.
6. Click OK. The parts are laid out in rows, biggest first, starting at the origin.
7. Down in the timeline, Arrange added one step named Arrange.
8. Double-click that step to change the layout later, or drag the marker back before it to see the parts where they were.
9. Now let's export for printing. Click the Utilities tab, then 3D Print.
10. It counts all five bodies. Pick three M F as the format. Bambu Studio and other slicers open it with every part in place.
11. Leave Refinement on Medium, and click OK. The file is saved, ready for your slicer.

That's it. Arrange packs your parts into neat rows, and 3D Print saves them together in one file.

LESSON 24 Video in Ironcast: Home › Learn › “Getting your part ready to 3D print” (1:50)

Getting your part ready to 3D print

Export your part as an STL or 3MF file, understand units, and know which file to open in a slicer like Bambu Studio.



In this lesson you'll get a part ready for your 3D printer.

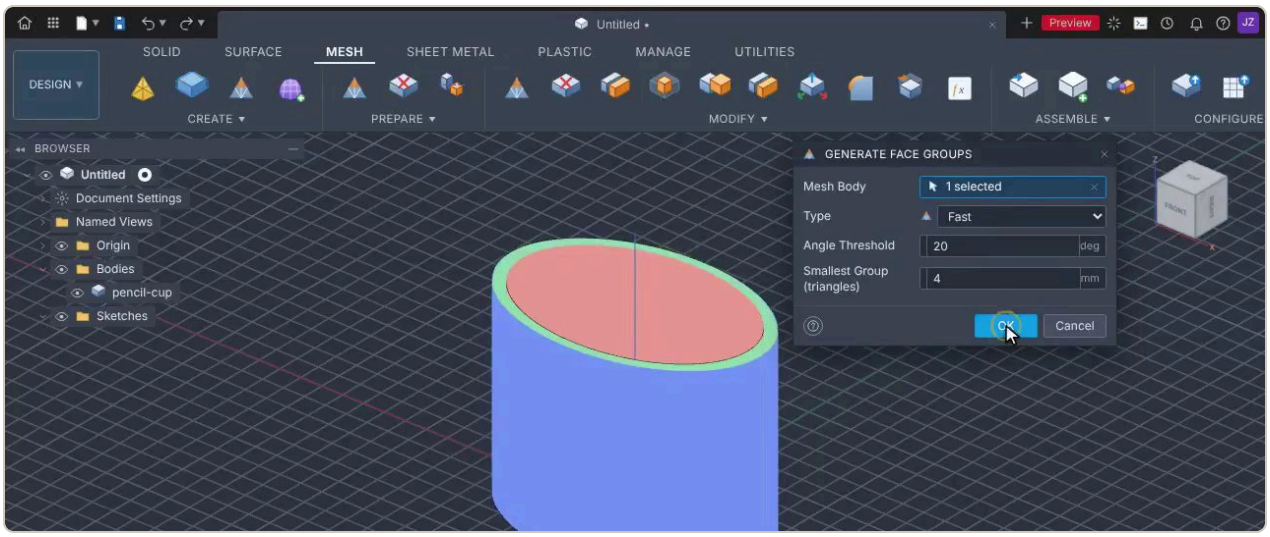
1. Here's a simple part, a small block with rounded corners and a hole through the top. It's 40 mm long, 30 wide, and 15 tall.
2. Before exporting, let's check the units. In the Browser, open Document Settings.
3. Ironcast works in millimeters, and that's what your exported file uses too. Slicers like Bambu Studio also expect millimeters, so the part opens at exactly the right size.
4. Now click the File menu in the top left.
5. Choose Export.
6. The Export window asks for a name and a file type. Give your part a name first.

7. For printing you want a mesh file, so open the Type list and choose STL.
8. The note at the bottom says it plainly. STL and 3MF are for your slicer.
9. Click Export.
10. Ironcast saves the file for you. That STL is the file you take to your slicer.
11. You can also export a 3MF. It's a newer format that Bambu Studio handles very well. Open File, Export again.
12. Pick 3MF from the Type list, and click Export.
13. There's also a shortcut. On the Utilities tab, in the Make group, click 3D Print.
14. It asks for a format, STL or 3MF, and says it saves a file your slicer can open. Click OK to save one more.
15. Now over to your slicer. In Bambu Studio, choose File, Import, and pick your STL or 3MF file. Your part appears at its real size, ready to slice.

LESSON 25 Video in Ironcast: Home › Learn › “Bring in a 3D-print file” (1:53)

Bring in a 3D-print file

Drop a 3MF print file into Ironcast as a mesh body, look at the Mesh tools, stretch it with Non Uniform Scale, check its walls with Thickness Analysis and split it into Face Groups.



In this lesson you'll bring a 3D printing file into Ironcast, change its size, and check it before you print.

1. Print files like 3MF and STL are meshes, made of tiny triangles. Insert Mesh, on the Mesh tab, opens one.
2. Or just drag the file from your folder and drop it onto the Ironcast window. Here's a pencil cup.
3. It comes in as a mesh body, and the browser shows it under Bodies, named after the file.
4. The Prepare and Modify groups hold the mesh tools, like repair, reduce and smooth.
5. Let's make the cup wider and taller. On the Solid tab, open the Modify menu and choose Scale.
6. Click the cup, then change Scale Type to Non Uniform. Now X, Y and Z each get their own number.

7. Set Point to Origin, so the cup stays on the ground. Set X Scale and Z Scale to 1.5, and click OK. Now it's oval, and taller.
8. Are the walls thick enough to print? On the Plastic tab, open the Inspect menu and choose Thickness Analysis.
9. Green means the walls are fine. The thinnest wall is 2 mm, well above the 1.2 minimum.
10. Raise Minimum Thickness to two and a half. The narrow sides turn red, too thin. The sides we stretched got thicker, so they're only yellow, close to the minimum.
11. Click OK. One more mesh tool: back on the Mesh tab, click Generate Face Groups, then click the cup.
12. Ironcast sorts the triangles into groups, each in its own color. Click OK.

That's it. Drop in a print file, adjust it, and check it before you send it to your printer.

CHAPTER 5

Surfaces, freeform shapes and meshes

Thin surface bodies, sculpted Form shapes, and repairing scanned or downloaded meshes.

Working with surfaces
Use the Surface tab: fill an open hole with Patch, then Stitch separate surfaces together into a solid.

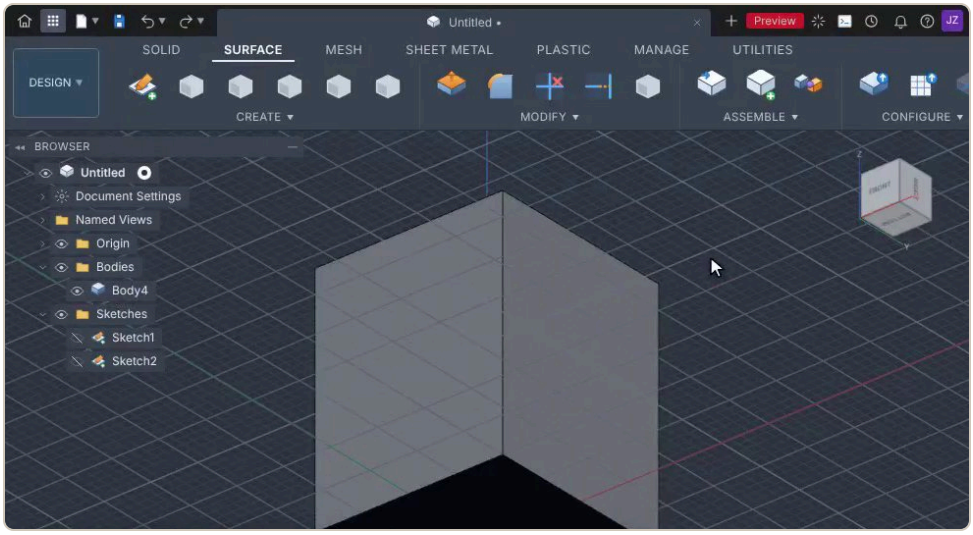
Form: shapes from sketches
Build smooth T-spline forms from sketches with Revolve, Pipe, Loft and Sweep, then Freeze vertices, edit, Make Uniform and Finish Form.

Form: editing tools
Mirror a Form box with Mirror - Internal, edit both sides, Clear Symmetry, Insert Point, UnWeld Edges, Convert a filleted body into a T-spline and reshape it, then Finish Form.

Fixing a mesh before you print
Turn a solid into a mesh, then use Repair, Reduce and Smooth to get it ready for a slicer, and learn why each step matters for 3D printing.

Working with surfaces

Use the Surface tab: fill an open hole with Patch, then Stitch separate surfaces together into a solid.

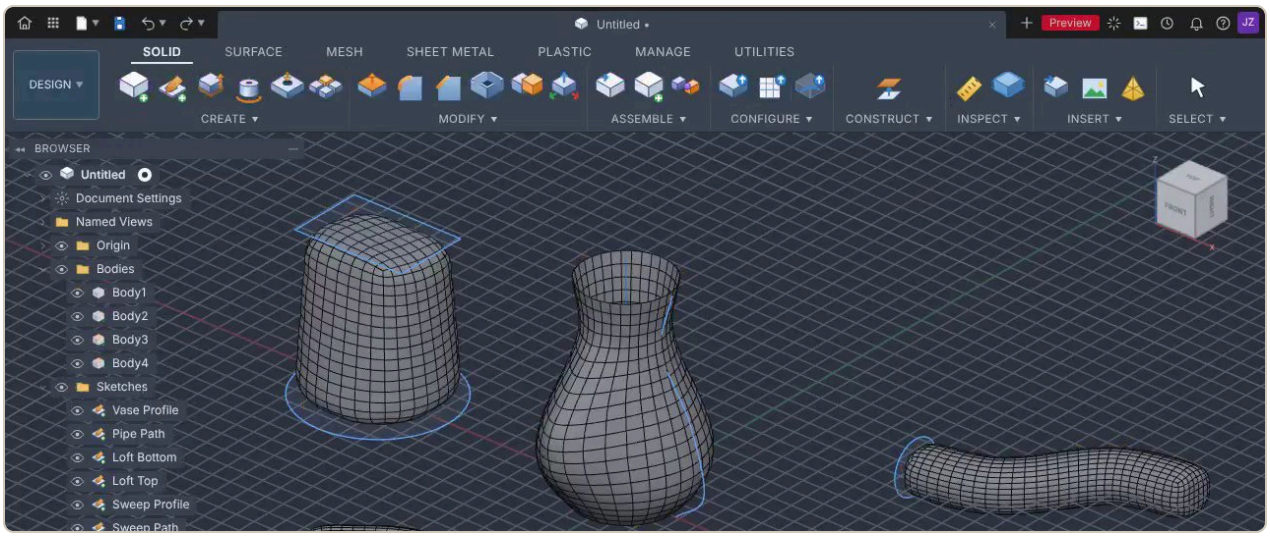


In this lesson you'll work with surfaces. They are thin skins with no thickness, and you can join them into a solid.

1. Here's a box built only from surfaces: four walls and a top. In the browser, surface bodies have their own icon.
2. Surface tools live on their own tab. Click Surface, next to the Solid tab.
3. Let's look at the box from underneath. There's no surface on the bottom, so it's open.
4. To close it, click Patch in the Create group. Patch fills an opening with a new surface.
5. Turn to look straight at the bottom, so the edges are easy to click.
6. Click one edge of the opening. With Chain Selection on, Ironcast selects all four edges around it at once.
7. The dialog says 4 edges are selected. Leave Continuity on Connected, and click OK.
8. A new surface body appears in the browser. Turn the box around, and the bottom is now covered.
9. But these are still three separate surfaces. To join them, click Stitch, in the Modify group.
10. Stitch asks for the surfaces to join. Click each of the three bodies in the browser.
11. Three are selected. The tolerance is how close the edges need to be to count as touching. The default is fine, so click OK.
12. The three surfaces are now one body, and its icon has changed from a surface to a solid. The skin is closed, so it holds volume.

Form: shapes from sketches

Build smooth T-spline forms from sketches with Revolve, Pipe, Loft and Sweep, then Freeze vertices, edit, Make Uniform and Finish Form.



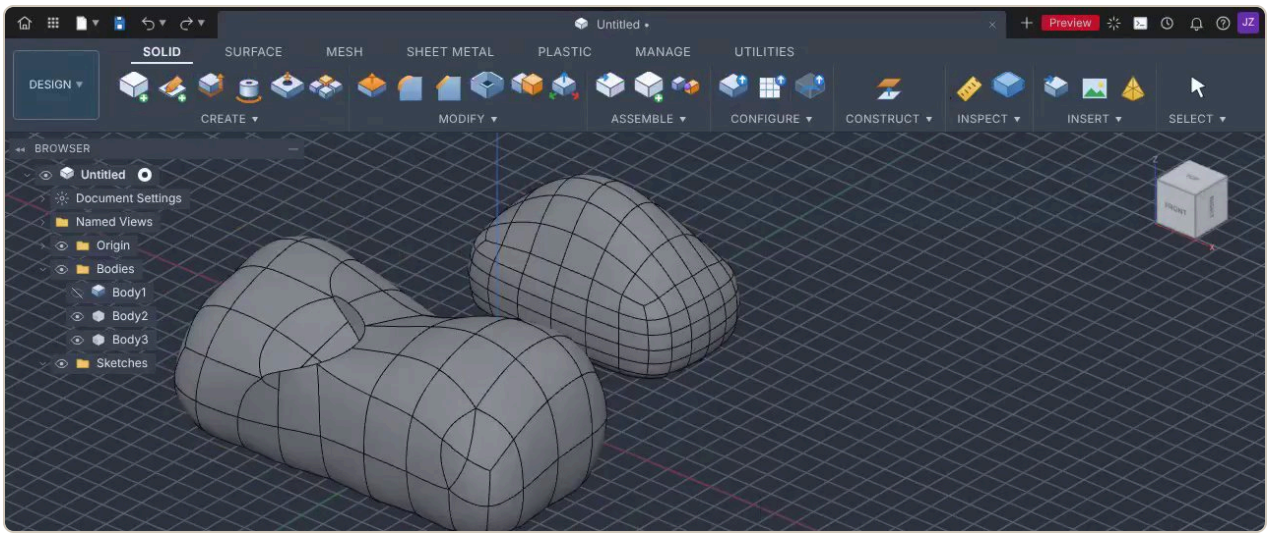
In this lesson you'll turn sketches into smooth Form shapes.

1. Here are the sketches: half a vase outline, a curvy path, a circle under a rectangle, and a small circle at the start of another path.
2. Click Create Form. The Form tab opens with a Box dialog. Click Cancel, since we'll use the sketches.
3. Open Create and choose Revolve. Pick the Vase Profile sketch, spin it around the Z axis, and set Segment Length to 12.
4. Click OK, and a smooth vase appears.
5. Next, choose Pipe, pick the Pipe Path sketch, and click OK. A tube follows the path.
6. Now Loft. Pick the circle first, then the rectangle, and click OK. It blends from round to square.
7. Then Sweep. Pick the small circle as the profile, the other curve as the path, and click OK.
8. To see the control points, open Utilities, Display Mode, and choose Control Frame Display. Then zoom in on the vase.
9. Hold Shift and click two vertices on the front of the vase. Then open Modify and choose Freeze. Frozen vertices turn blue.
10. Click Edit Form, set the Selection Filter to Face, and click the face just left of the blue vertices. Type −15 for Y Distance.
11. The free side of the face pushes out, but the two blue vertices stay right where they are. Click OK.
12. Open Utilities and choose Make Uniform. Every face is split evenly, for more control points.
13. Click Finish Form, and each form becomes a solid body in the browser.

That's it. Sketch a profile or a path, and Form turns it into a smooth shape.

Form: editing tools

Mirror a Form box with Mirror - Internal, edit both sides, Clear Symmetry, Insert Point, UnWeld Edges, Convert a filleted body into a T-spline and reshape it, then Finish Form.



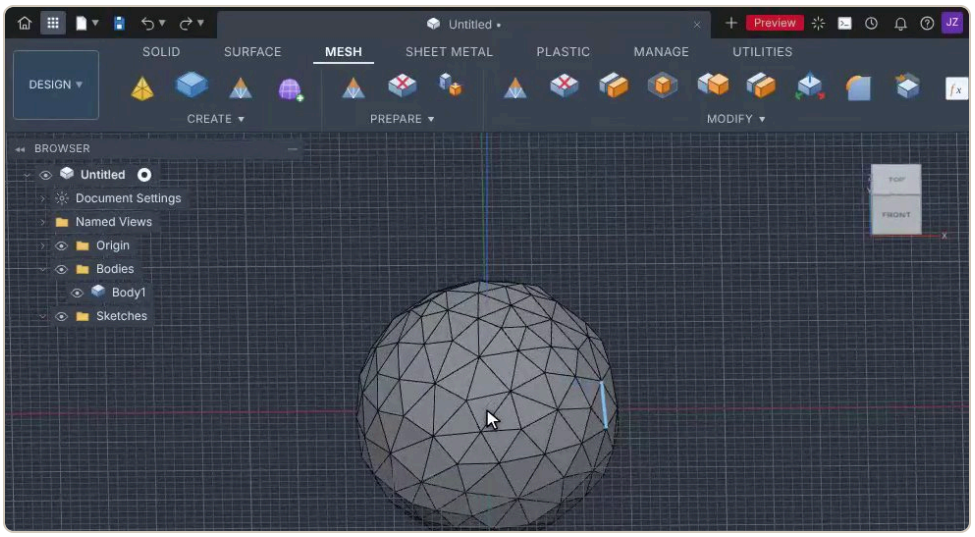
In this lesson you'll use the Form editing tools: symmetry, Insert Point, UnWeld Edges, and Convert.

1. Here's a rounded block we'll use later. Click Create Form, and the Box dialog opens.
2. Set Length to 40, X to 20, and Y to −30, so the box sits right of the center line. Click OK.
3. Open Utilities, Display Mode, and choose Control Frame Display to see the control points.
4. Open Symmetry, choose Mirror - Internal, and click OK. The left half appears, joined at the seam.
5. Click Edit Form and set the filter to Face. Click the top face at the right end, then Shift-click its twin on the left.
6. Type 12 for Z Distance. Both ends rise the same amount. Click OK.
7. Open Symmetry and choose Clear Symmetry. Each side now edits on its own.
8. Click the front top edge on the right, then choose Modify, Insert Point. A new point appears in the middle of the edge.
9. Now click the seam edge at the front of the top, and choose Modify, UnWeld Edges.
10. Click Edit Form, pick the Face filter, and click the face right of that edge. Type eight for Z. The face pulls away and opens a gap.
11. Click OK. Next, open Utilities and choose Convert. Body1 is picked, with three faces per side.
12. Click OK. The block is wrapped in a smooth T-spline cage.
13. Click Edit Form, click a face on top of the block, and type 12 for Z. The top swells smoothly. Click OK.
14. Click Finish Form, and both forms become solid bodies.

That's it. Mirror half a form for symmetry, add points and seams where you need them, and Convert bodies to sculpt them.

Fixing a mesh before you print

Turn a solid into a mesh, then use Repair, Reduce and Smooth to get it ready for a slicer, and learn why each step matters for 3D printing.



In this lesson you'll get a mesh ready for 3D printing, using Repair, Reduce and Smooth.

1. Most files you download or scan are meshes, surfaces made of thousands of tiny triangles. Here we have a smooth ball, and we'll turn it into a mesh to practice on.
2. Click the Mesh tab at the top. It holds all the tools for working with triangles.
3. In the Create group, click Tessellate. This chops a solid into triangles.
4. Click the ball to pick it. Then set Refinement to Low, so we get a lighter mesh, and click OK.
5. Now you can see every triangle as a black line. This is what a slicer sees when you send it a mesh.
6. First, repair it. In the Prepare group, click Repair, then click the mesh.
7. Repair Type is set to Close Holes. Printers need a mesh with no gaps and no loose seams, a sealed shell, so the slicer can tell what is inside and what is outside. Click OK.
8. Next, the mesh may be heavier than it needs to be. In the Modify group, click Reduce, and click the mesh.
9. The panel tells you how many triangles there are right now. Too many makes big files and a slow slicer.
10. Type 15 in Proportion to keep just 15% of the triangles. Watch the preview get coarser, then click OK.
11. Fewer triangles means a smaller file, but too few makes the surface look blocky. So reduce a little at a time.
12. Last, Smooth. It's in the Modify group. Click it, then click the mesh.
13. Smoothness decides how strongly lumps and rough spots get relaxed. Type 80, and click OK.

14. A smoother surface prints with fewer rough patches. Turn the model around to check it from every side.

That's the routine: repair the holes, reduce the weight, smooth the rough spots, and your mesh is ready for the slicer.

CHAPTER 6

Keeping a design organised

Names, groups and tools for checking what you have built.

Organize: timeline groups and selection sets
Group timeline features into a folder you can fold up, save a selection as a Selection Set, and reuse it with Fill Hole.

Body, sketch and timeline groups
Tidy the browser with body groups and sketch groups: make a group from selected bodies, move another body in, rename it and hide it in one click. Then group two timeline features and rename the group.

Measure and section view
Use Measure to check distances between corners, then cut the part open with Section Analysis to look inside.

Inspect: surface analysis
Check a rounded part with Zebra, Curvature Map, Minimum Radius, Environment Map and Accessibility analysis from the Inspect menu.

Materials and mass
Give a part a physical material, add your own material with Manage Materials, and watch the mass change in Properties.

LESSON 30 — Video in Ironcast: Home › Learn › “Organize: timeline groups and selection sets” (1:37)

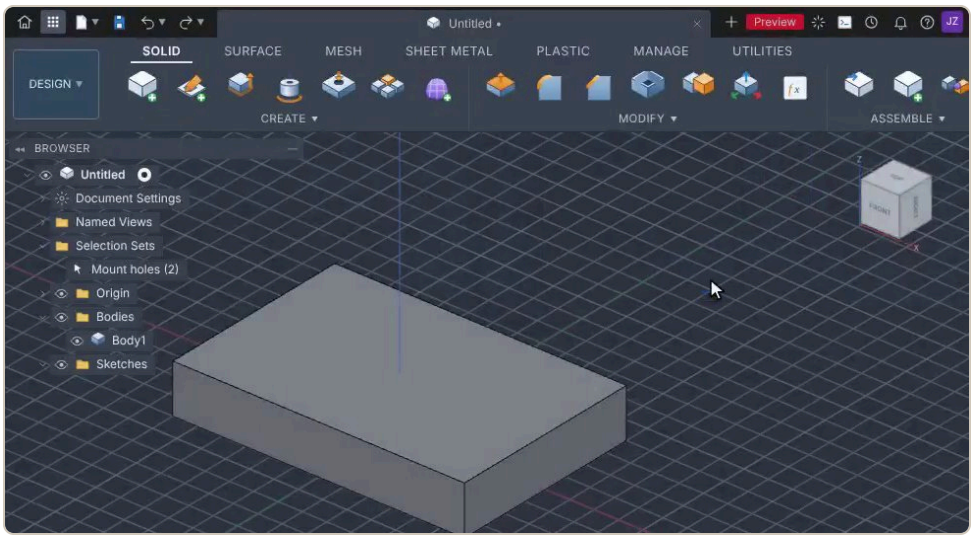
Organize: timeline groups and selection sets

Group timeline features into a folder you can fold up, save a selection as a Selection Set, and reuse it with Fill Hole.

In this lesson you'll keep your design tidy with timeline groups and selection sets.

1. Here's a plate with two holes. Down at the bottom, the timeline shows each step: the box, then the two holes.

2. Let's group the two holes. Click the first Hole in the timeline, then hold Shift and click the second one.

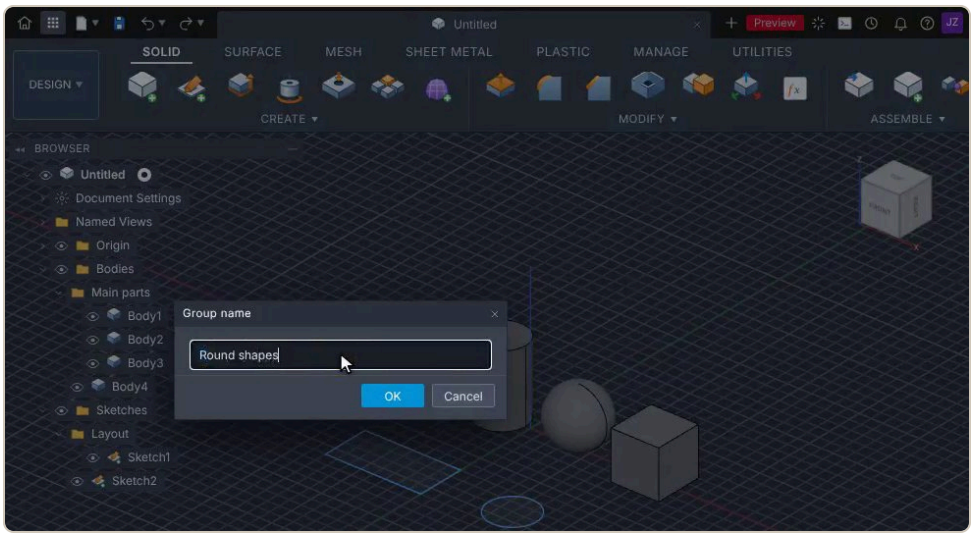


3. Right click one of them, and choose Create Group.
4. A folder chip called Group1 appears in front of them. Click it, and the group folds up into one chip. That's handy when the timeline gets long.
5. Click the folder again to open the group back up.
6. Now, selection sets. Click the inside wall of the first hole. Then hold Shift, and click the inside wall of the second hole.
7. Right click one of the walls, and choose Create Selection Set.
8. Give it a name, like Mount holes, and press Enter.
9. The set is saved in the browser, under Selection Sets. The two means it holds two faces.
10. Click in empty space to clear the selection. Later, just click the set in the browser, and both walls are picked again in one go.
11. Let's use it. Open the Modify menu, and choose Fill Hole. Both hole walls are already in the list.
12. Click OK, and both holes are filled back to solid. That's great for cleaning up downloaded parts.

That's it. Groups keep the timeline short, and selection sets save you from picking the same things again and again.

Body, sketch and timeline groups

Tidy the browser with body groups and sketch groups: make a group from selected bodies, move another body in, rename it and hide it in one click. Then group two timeline features and rename the group.



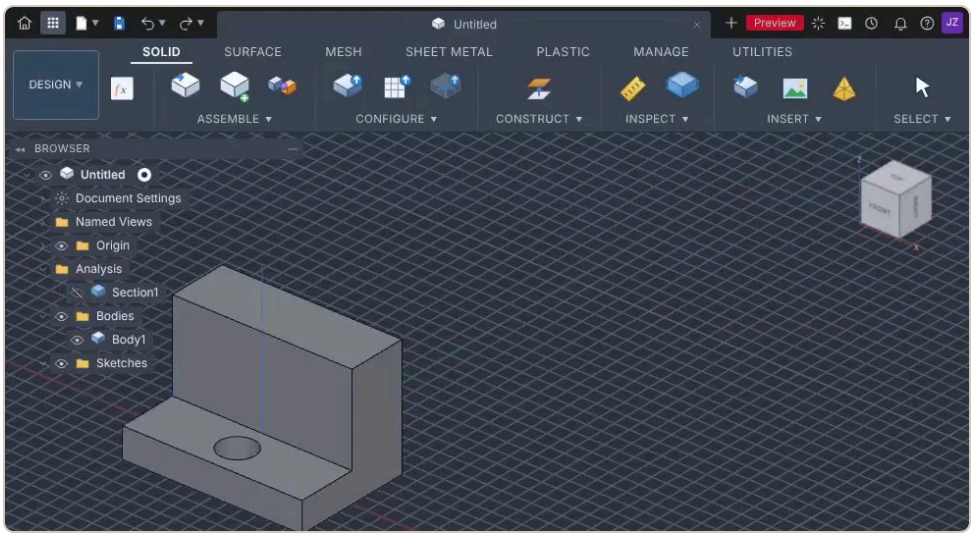
In this lesson you'll tidy up the browser and the timeline with groups.

1. Here are four bodies and two sketches. In the browser, they're listed under Bodies and Sketches.
2. Click Body1, then hold Control and click Body2, to select the block and the cylinder.
3. Right click the Bodies folder, and choose New Body Group.
4. Name it Left parts, and click OK. The two bodies now sit inside the new group.
5. To add another body, right click Body3, the sphere, choose Move to Group, and pick Left parts.
6. Now right click the group, and choose Rename. Call it Main parts, and click OK.
7. Click in empty space to clear the selection. Then right click the group, and choose Show Hide. All three bodies disappear at once.
8. Do the same again, and they come back.
9. Sketches work the same way. Right click the Sketches folder, choose New Sketch Group, and name it Layout.
10. Right click Sketch1, choose Move to Group, and pick Layout. Now the sketch lives in that folder.
11. The timeline can have groups too. Click the cylinder in the timeline, then hold Shift and click the sphere.
12. Right click one of them, and choose Create Group. A folder chip appears in front of them.
13. Right click the group chip, choose Rename, type Round shapes, and click OK.

That's it. Groups keep big designs easy to find your way around.

Measure and section view

Use Measure to check distances between corners, then cut the part open with Section Analysis to look inside.



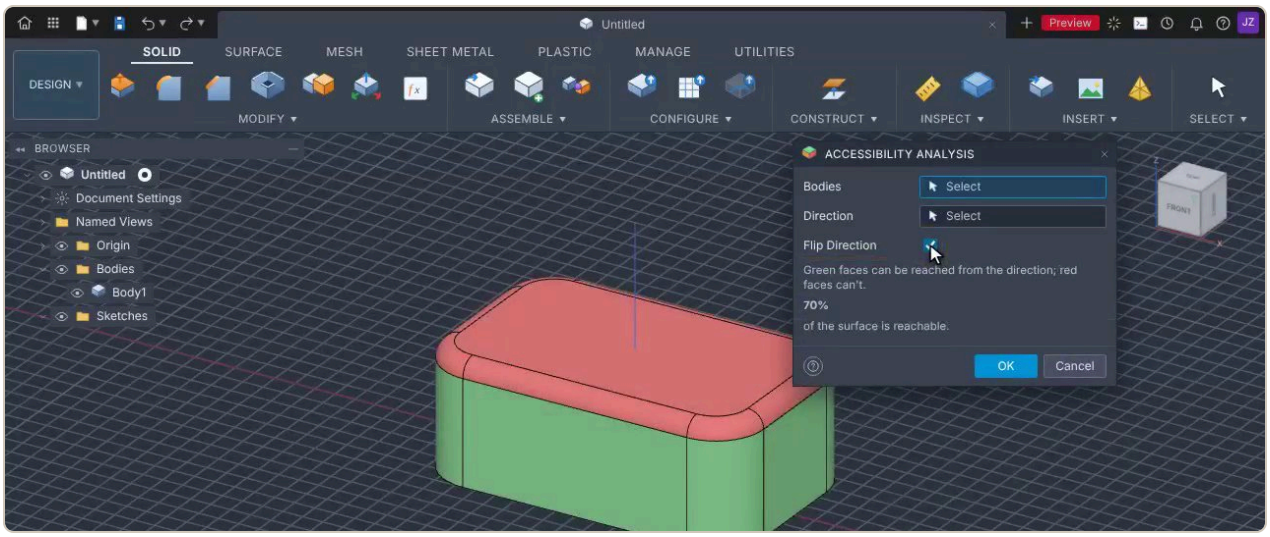
In this lesson you'll measure a part, and then cut it open to see inside.

1. Here's an L shaped bracket with a hole in its base. Let's check how big it is.
2. Press the I key to open Measure.
3. Click the top corner of the upright wall, and then the corner at the bottom of the wall.
4. The panel shows the distance, 30 mm, and how far apart the points are in X, Y and Z.
5. Now click the front corner of the base. Measure always uses the last two points you clicked, so the base sticks out 20 mm.
6. Want inches too? Set Secondary Units to Inch, and each distance also shows in inches.
7. Click Close. Next, let's look inside the part.
8. Slide the toolbar over to the Inspect group, and click Section Analysis.
9. Click the flat side face of the bracket. A blue plane shows where the cut starts.
10. In Offset, type −30. That slides the cut to the middle of the part, and now you can see the hole cut in half.
11. Turn on Flip to keep the other half instead.
12. Click OK. The cut stays on, and a new Analysis folder appears in the browser.
13. Open the folder, and click the eye next to Section one. The section turns off, and the whole part is back.

That's it. Measure checks sizes, and Section Analysis lets you see inside any part.

Inspect: surface analysis

Check a rounded part with Zebra, Curvature Map, Minimum Radius, Environment Map and Accessibility analysis from the Inspect menu.



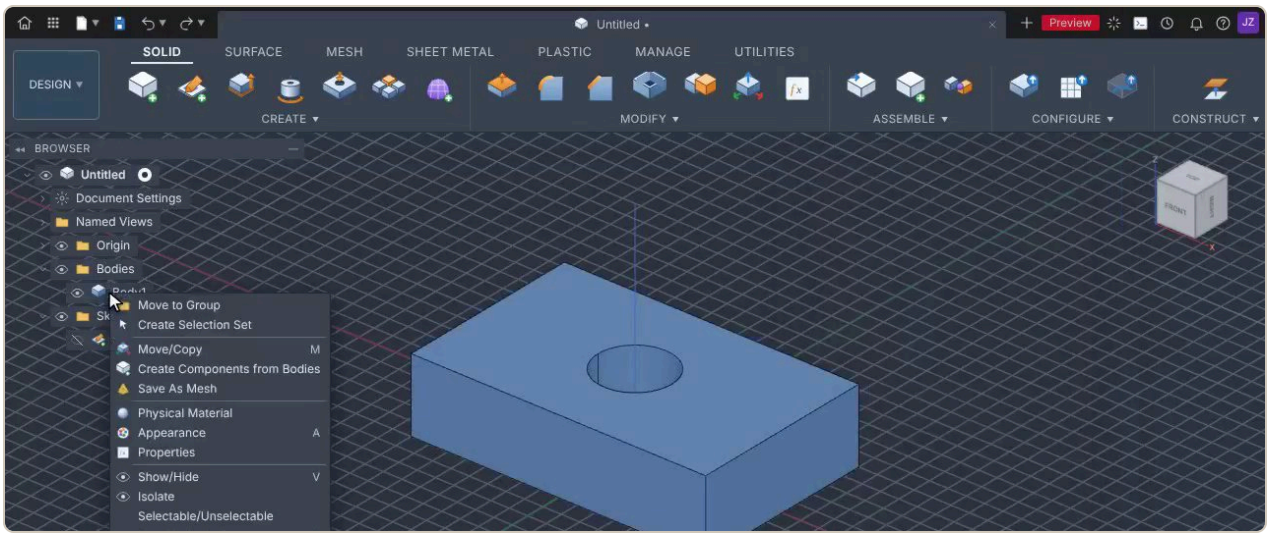
In this lesson you'll use the Inspect tools that color a part, to check how smooth and how tight its curves are.

1. Here's a box with rounded corners: 12 mm fillets up the sides, and 4 mm fillets around the top.
2. Slide the toolbar over, open the Inspect menu, and choose Zebra Analysis.
3. Stripes reflect on the part, like light on a shiny car. Set Stripe Density to three for wider stripes. Smooth stripes mean smooth surfaces.
4. Click Cancel, open Inspect again, and choose Curvature Map Analysis.
5. Flat faces are blue. The big side fillets show orange, and the small top fillets are red, because they're tighter than 10 mm.
6. Cancel, and choose Minimum Radius Analysis. Nothing is red yet, because no curve is tighter than 1 mm.
7. Type five in Minimum Radius. The 4 mm fillets turn red: a 5 mm cutter couldn't make them.
8. Type 15, and the 12 mm side fillets turn red too.
9. Cancel, and choose Environment Map Analysis. The part becomes a mirror of the sky, the horizon and the floor.
10. Orbit around, and watch the reflections slide smoothly over the rounded edges.
11. Cancel, and choose Accessibility Analysis. Green faces can be reached from straight above, like a cutter coming down.
12. Turn on Flip Direction to look from below instead. Now the top faces turn red, because they face away.

That's it. All these checks live in the Inspect menu, ready whenever you want to test a shape.

Materials and mass

Give a part a physical material, add your own material with Manage Materials, and watch the mass change in Properties.



In this lesson you'll give a part a real material, add a material of your own, and see how much the part weighs.

1. Here's a block, 80 × 50 × 20 mm, with a hole through the middle.
2. First, how heavy is it? Right-click Body one in the browser, and choose Properties.
3. No material is set yet, so Ironcast uses steel. The mass is about 579 grams.
4. Close the window. Click the block to select it, then open the Modify menu and choose Physical Material.
5. Click Aluminum in the library. Aluminum now shows under In This Design, so you can see what this part is made of.
6. Click Close, and open Properties again. Aluminum is much lighter: about 199 grams.
7. What if your material isn't in the list? Close this, open the Modify menu, and choose Manage Materials.
8. Type Lead as the new material, and 11.34 for its density, in grams per cubic centimeter.
9. Click Add. Lead now shows in your own materials, and it stays on this computer for every design.
10. Click Close. Right-click Body one, and choose Physical Material again.
11. Scroll to the bottom of the library. There's Lead, at 11.34. Click it.
12. Click Close, and open Properties one more time.
13. Now it's Lead, and the same block weighs about 836 grams. Four times the aluminum one.

That's it. Pick a physical material, or add your own, and Properties always shows what your part really weighs.

CHAPTER 7

Assemblies, looks and motion

Parts that move together, appearances, rendered pictures and exploded animations.

Components and joints
Turn two bodies into components, connect them with a Revolute joint, and swing the joint with Drive Joints.

Contact, fasteners and sustainability
Make a sliding block with a Slider joint, turn on Enable All Contact so it stops at a wall, check a screw with Fastener Stack Analysis, and estimate embodied carbon with Sustainability Insights.

Appearance and materials
Color bodies with the Appearance library, set a Physical Material, and see the part with real lighting in the Render workspace.

Render: in-canvas render and saving a picture
Give a part its appearances, switch to the Render workspace, turn on In-canvas Render and its quality setting, then render a picture and save it from the Rendering Gallery.

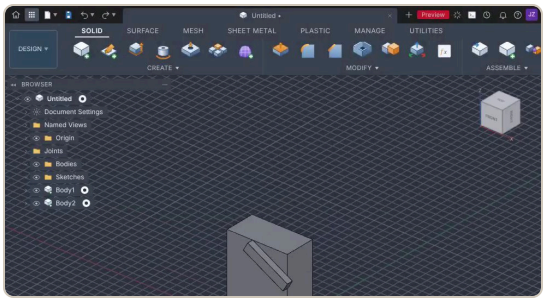
Animation: storyboards
Animate a three-part assembly in the Animation workspace: Transform Components, Auto Explode, a camera View action, then Play and Publish Video.

LESSON 35

Video in Ironcast: Home › Learn › “Components and joints” (1:37)

Components and joints

Turn two bodies into components, connect them with a Revolute joint, and swing the joint with Drive Joints.



In this lesson you'll connect two parts with a joint, so one of them can move.

1. Here are two separate bodies: a tall block and a thin bar. We'll hang the bar on the front of the block, like a pendulum.

2. Joints work between components, so first we turn each body into a component. Open the Assemble menu, and choose Create Components from Bodies.
3. Now each body is its own component. You can see them in the browser, with a component icon next to each name.
4. Next, open Assemble again and click Joint.
5. For Component 1, click the front of the block, near the top. That's where the pivot will be.
6. Then click the Component 2 box, and click the front of the bar, near its bottom end.
7. The bar flips around and jumps onto the block, so the two spots you clicked meet. Now it hangs from the block.
8. Right now the type is Rigid, which glues the parts together. Change Type to Revolute, so the bar can turn around the pivot.
9. Click OK. The new joint is listed in the browser, in the Joints folder.
10. To make it move, open Assemble and choose Drive Joints.
11. Drag the Rotation slider. The bar swings around the pivot, just like a real hinge.
12. Click Animate, then OK, and watch Ironcast swing the bar back and forth for you.

That's it. Make components, join them with a Revolute joint, and drive the joint to test how your parts move.

LESSON 36 Video in Ironcast: Home › Learn › “Contact, fasteners and sustainability” (1:42)

Contact, fasteners and sustainability

Make a sliding block with a Slider joint, turn on Enable All Contact so it stops at a wall, check a screw with Fastener Stack Analysis, and estimate embodied carbon with Sustainability Insights.



In this lesson you'll make parts stop when they touch, check a screw, and estimate carbon.

1. Here's a base plate with a wall, and a separate block that we'll make slide.

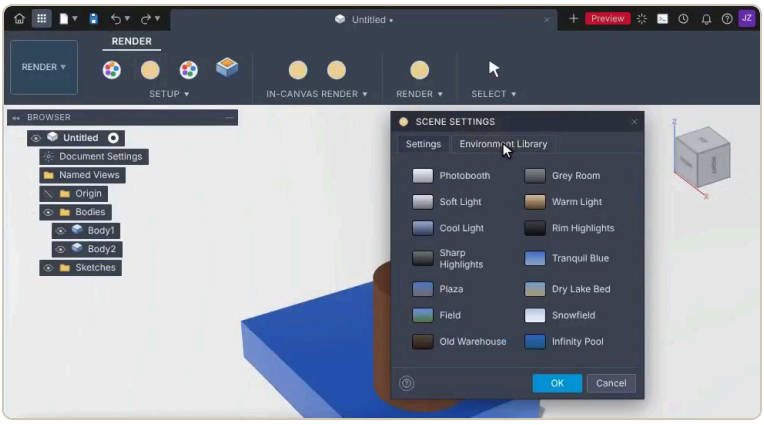
2. Open Assemble, and choose Create Components from Bodies.
3. Open Assemble again, and click Joint. For Component 1, click the wall's inside face.
4. Click the Component 2 box, then the end of the block. It jumps against the wall.
5. Change Type to Slider, and click OK.
6. Open Assemble, choose Drive Joints, and drag the slider. The block slides away.
7. Drag it back, and it goes right through the wall.
8. Slide it back out, and click OK.
9. Now open Assemble, and choose Enable All Contact.
10. Open Drive Joints again, and drag the block toward the wall.
11. This time it stops at the wall, and the status bar says contact. Click OK.
12. Now a screw. Open Insert, choose Insert Fastener, and click the hole's edge.
13. Set Size to M5, and Length to 12. Click OK.
14. Open Inspect, and choose Fastener Stack Analysis.
15. The plate is 6 mm thick, so the screw sticks out six past it. Ironcast suggests a nut, or a shorter screw.
16. Last, switch to the Utilities tab, and click Sustainability Insights.
17. It lists each body's mass, and estimates the carbon it took to make.

That's it: contact, fastener checks, and sustainability insights.

LESSON 37 Video in Ironcast: Home › Learn › “Appearance and materials” (1:37)

Appearance and materials

Color bodies with the Appearance library, set a Physical Material, and see the part with real lighting in the Render workspace.



In this lesson you'll give your parts colors and materials, and see them with real lighting.

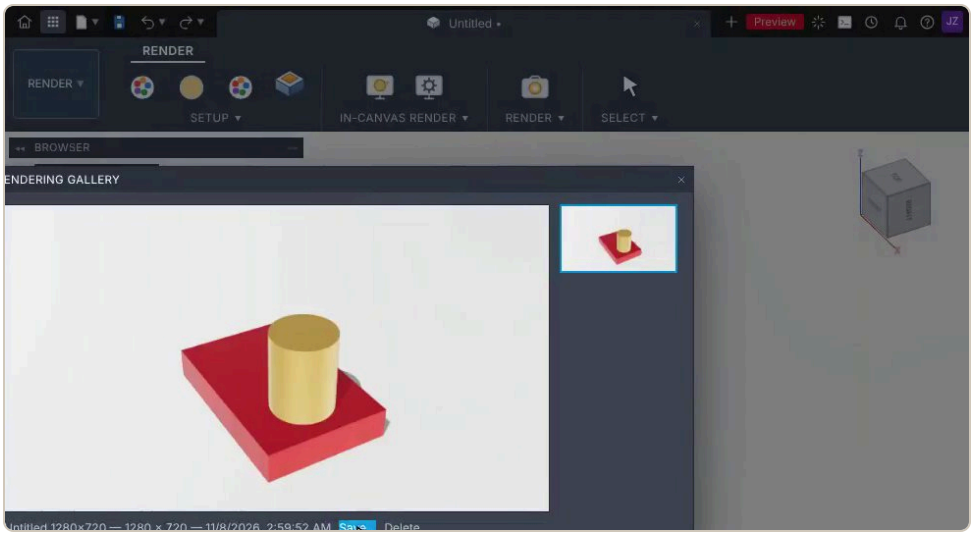
1. Here's a simple part made of two bodies, a flat base and a round post. Right now they're both plain gray.
2. Press the A key to open the Appearance panel.

3. Click the base to select it. The Selection box now says one selected.
4. The library is sorted into groups. Click Metal to see only the metals.
5. Click Aluminum, Anodized Blue. The base turns blue right away, and the color shows up under In This Design.
6. Now the post. Click the base again to unselect it, then click the post.
7. Scroll down to the Wood group, click it, and pick Walnut. Now the post looks like wood.
8. Click Close when you're done.
9. Appearance only changes the look. To set what a body is really made of, right click it in the browser, and choose Physical Material.
10. Pick Aluminum from the list. Ironcast uses this to work out the real weight of your part.
11. Finally, the render look. Click Design at the top left, and switch to the Render workspace.
12. Now your part is shown with realistic lighting and shiny reflections, on a bright studio background.
13. Scene Settings changes the lighting. Open the Environment Library tab, and try a different light, like Warm Light.
14. Click OK to keep it. That's appearance, materials and the render look. Have fun making your parts look real.

LESSON 38 Video in Ironcast: Home › Learn › “Render: in-canvas render and saving a picture” (1:40)

Render: in-canvas render and saving a picture

Give a part its appearances, switch to the Render workspace, turn on In-canvas Render and its quality setting, then render a picture and save it from the Rendering Gallery.



In this lesson you'll make a realistic picture of your part with the Render workspace.

1. First, give the part some color. Press the A key to open Appearance, and click the base to select it.
2. Click Metal, then Aluminum, Anodized Red. The base turns red.
3. Click the base again to unselect it, click the post, and pick Brass, Satin.

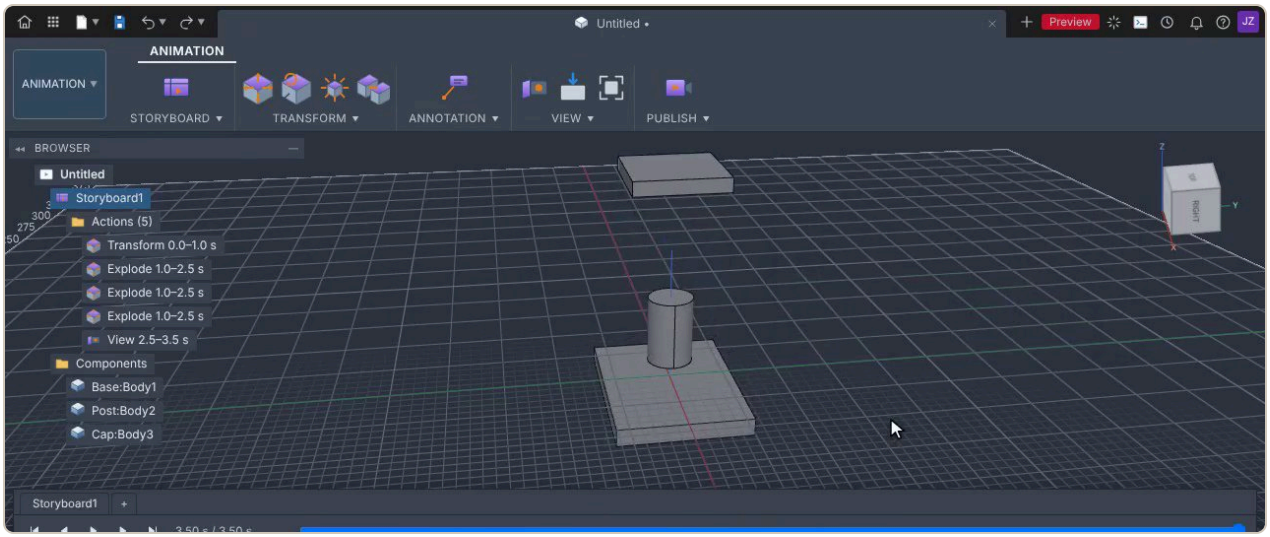
4. Click Close. Now click Design at the top left, and switch to the Render workspace.
5. The part now shows with studio lighting, shadows and shiny metal, on a bright background.
6. Next, click In-canvas Render. It keeps this realistic look on, even in the other workspaces.
7. To see it, switch back to Design. The part still looks real while you keep working on it.
8. Go back to the Render workspace. Next to In-canvas Render is its settings button.
9. Click it. Render Quality can be Fast, Standard, or Excellent for smoother edges. Choose Excellent, and click OK.
10. Now let's save a picture. Click Render, the camera button.
11. Pick the image size, like 92 × 27, and the quality. You can also give it a see-through background, and choose PNG or JPEG.
12. Click Render. Ironcast renders the picture on your computer, and opens it in the Rendering Gallery.
13. Click Save, and the picture is saved as an image file you can share.

That's it: appearances, in-canvas render, and a saved picture of your design.

LESSON 39 Video in Ironcast: Home › Learn › “Animation: storyboards” (1:58)

Animation: storyboards

Animate a three-part assembly in the Animation workspace: Transform Components, Auto Explode, a camera View action, then Play and Publish Video.



Let's animate a small assembly.

1. It has three components: a base, a post, and a cap.
2. Click Design at the top left and switch to Animation. The storyboard timeline opens along the bottom.
3. Click the cap, then open Transform and choose Transform Components.
4. Set Z Distance to 40, and click OK. The move takes one second, starting at zero.

5. A Transform bar appears in the timeline. Click it, and the playhead jumps to its end. The cap has lifted off the post.
6. Now open Transform and choose Auto Explode, One Level. Set the scale to 0.3, and click OK.
7. Click one of the new Explode bars. Every part has moved away from the center.
8. Turn the view to a nicer angle. Then open View and choose Create View Action.
9. A Camera row appears. Now turn the view back the other way, so we can watch the camera glide to the saved view.
10. Click Go to Start, then Play, and watch the whole storyboard.
11. The browser lists every action with its start and end time. Click one to jump the playhead there.
12. To share it, open Publish and choose Publish Video. Give it a file name and frame rate, and click OK.
13. The storyboard plays once while Ironcast records it, then saves it as a web m video.

That's it. Add actions along the timeline, play them back, and publish your video.

CHAPTER 8

Sheet metal

Flanges and bends, flat patterns for cutting, and rules that keep a design consistent.

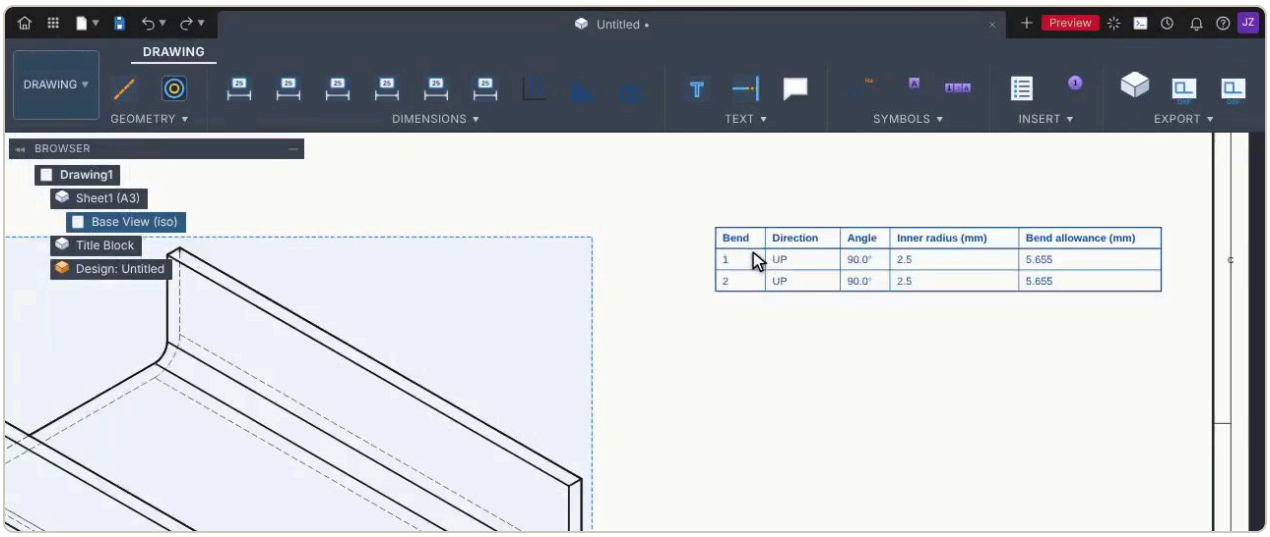
Sheet metal: flanges, flat pattern and bend table
Make a sheet from a sketch with Flange, add edge flanges to fold it into a channel, unfold it with Flat Pattern, and list the bends in a drawing with Bend Table.

More sheet metal
Pick a Sheet Metal Rule, make a sheet with Flange, add an edge flange and a Hem, Bend along a sketch line, Unfold and Refold, then Create Flat Pattern and export it as DXF.

Flatten, Navigator and configuration rules
Unroll a cylinder's side into a flat sketch with Flatten, search, zoom to and isolate bodies in the Navigator, then build a configuration table with a rule that hides a body when a parameter gets too big.

Sheet metal: flanges, flat pattern and bend table

Make a sheet from a sketch with Flange, add edge flanges to fold it into a channel, unfold it with Flat Pattern, and list the bends in a drawing with Bend Table.

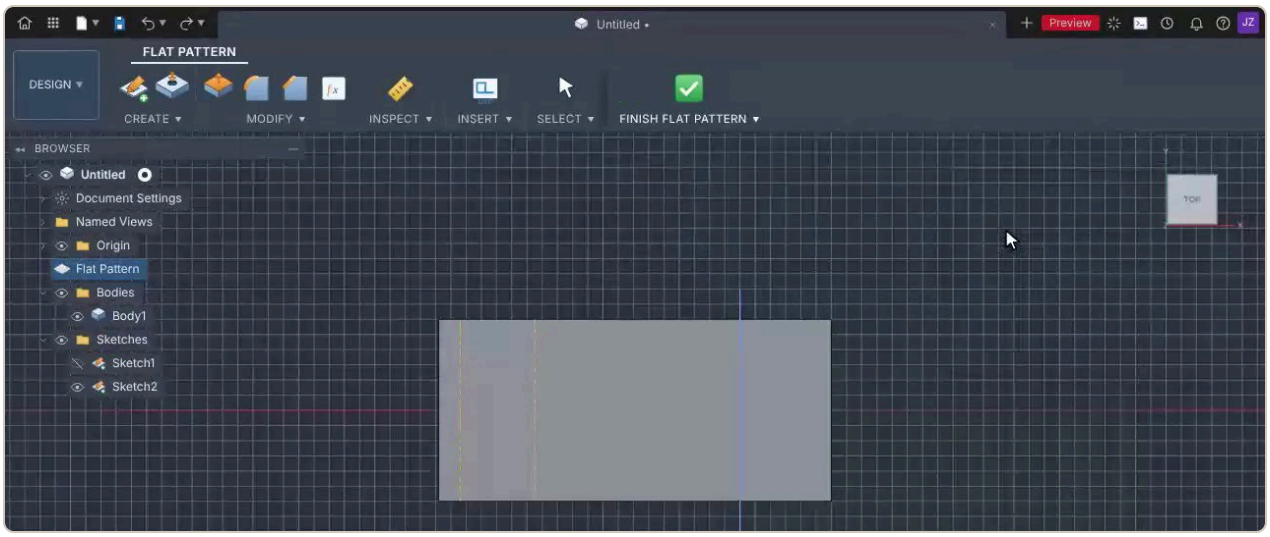


In this lesson you'll fold a part from sheet metal, unfold it flat, and list its bends on a drawing.

1. We start with a plain rectangle sketch, 80 × 50. The sheet metal tools are on the Sheet Metal tab.
2. Click Flange, then click inside the rectangle.
3. Ironcast turns it into a flat sheet. The thickness comes from the sheet metal rule, aluminum, two and a half millimeters. Click OK.
4. Now let's fold up the sides. Click the front top edge of the sheet, then hold Control and click the back edge.
5. Click Flange again. A flange folds up from both edges.
6. Set the Height to 20, and click OK. Now it's a U shaped channel with two bends.
7. To see the flat shape you'd cut from a sheet, click Create Flat Pattern, then click the floor of the channel, and click OK.
8. Ironcast unfolds the part flat, with the bends stretched out. When you're done looking, click Finish Flat Pattern.
9. Now a drawing. Click Design at the top left, point to Drawing, and choose From Design.
10. Set Sheet Scale to two to one, so the part is drawn twice as big. Click OK.
11. Change Orientation to Isometric. Click on the sheet to place the view, then click OK.
12. Open the Insert menu and choose Bend Table. Then click on the sheet where the table goes.
13. Zoom in for a closer look. There's one row for each bend, with its direction, angle and inner radius.
14. The last column, bend allowance, is how much flat sheet each bend uses up. That's sheet metal: flange, fold, flatten, and document it.

More sheet metal

Pick a Sheet Metal Rule, make a sheet with Flange, add an edge flange and a Hem, Bend along a sketch line, Unfold and Refold, then Create Flat Pattern and export it as DXF.

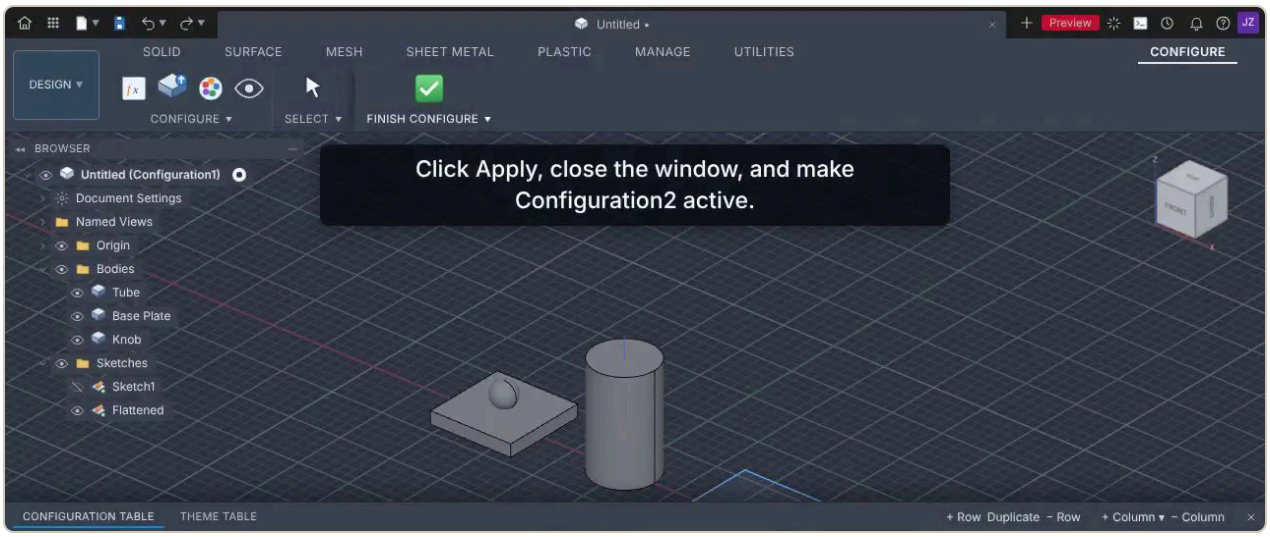


More sheet metal: rules, hems, bends and flat patterns.

1. On the Sheet Metal tab, open Modify, and choose Sheet Metal Rules.
2. Pick Steel, 3 mm thick. Set it as the active rule, and click OK.
3. Choose Flange, click inside the sketch, and click OK. Now it's a flat steel sheet.
4. Click the left edge, choose Flange again, and set Height to 25.
5. Click OK, and a wall folds up.
6. Click the top of the wall, choose Hem, and click OK. The edge folds back on itself.
7. For a bend, sketch a line on the sheet, running past both edges.
8. Choose Bend, click the line, and check Flip Side, so the short end moves.
9. Click OK. The end bends up 90°.
10. Unfold lays every bend flat.
11. And Refold folds it back up.
12. Create Flat Pattern shows the flat blank you'd cut.
13. Export it as a DXF for the cutter, then click Finish Flat Pattern.

Flatten, Navigator and configuration rules

Unroll a cylinder's side into a flat sketch with Flatten, search, zoom to and isolate bodies in the Navigator, then build a configuration table with a rule that hides a body when a parameter gets too big.



In this lesson you'll flatten a round face, find bodies with the Navigator, and add a configuration rule.

1. Here's a tube, made from a sketched circle and an extrude, next to a base plate with a knob.
2. Click the Surface tab, open Create, and choose Flatten. Then click the side of the tube.
3. The round side unrolls into a flat outline sketch, 20 mm from the model. Click OK.
4. Back on the Solid tab, open Inspect and choose Navigator. It lists every body in the design.
5. Type k n in the search box, and only the knob is left. Click Zoom To.
6. Click Isolate to hide everything else. Clear the search, and the other bodies show Hidden.
7. Click Show All to bring them back, and close the Navigator.
8. Now configurations. A parameter named height drives the tube. Open Configure, and choose Configure.
9. Click Configure Parameter and pick height. Then Configure Visibility, and pick Knob.
10. Click plus Row to add Configuration2, and set its height to 80.
11. Open Configure, Create Configuration Rules, and click plus Rule. When height is more than 60, set Knob to Hidden.
12. Click Apply, close the window, and make Configuration2 active.
13. The tube grows to 80, and the rule hides the knob. Configuration1 still keeps it.

CHAPTER 9

Drawings and 2D drafting

Dimensioned sheets from your 3D model, and AutoCAD-style 2D drawing that opens and saves DWG.

Make a 2D drawing
Turn a part into a 2D drawing: place a front view and projected views, add dimensions, a weld symbol and a revision table, then save it as a PDF.

Drawing extras: center marks, breaks and images
Add a Center Mark Pattern to a bolt circle, shorten a long view with Break View, dimension across the break, and insert a logo image.

Drawing a flat pattern
Put a sheet metal part's Flat Pattern on a drawing, label its bends with Bend Identifier, add Edge Extensions at a bend, clean up a centre mark with the Delete key, and output a PDF.

2D Drafting and DWG files
Draw a plate with typed commands the way you would in AutoCAD: rectangle, circle, fillet, hatch and a dimension. Then save it as a real DWG file.

LESSON 43

Video in Ironcast: Home › Learn › “Make a 2D drawing” (1:54)

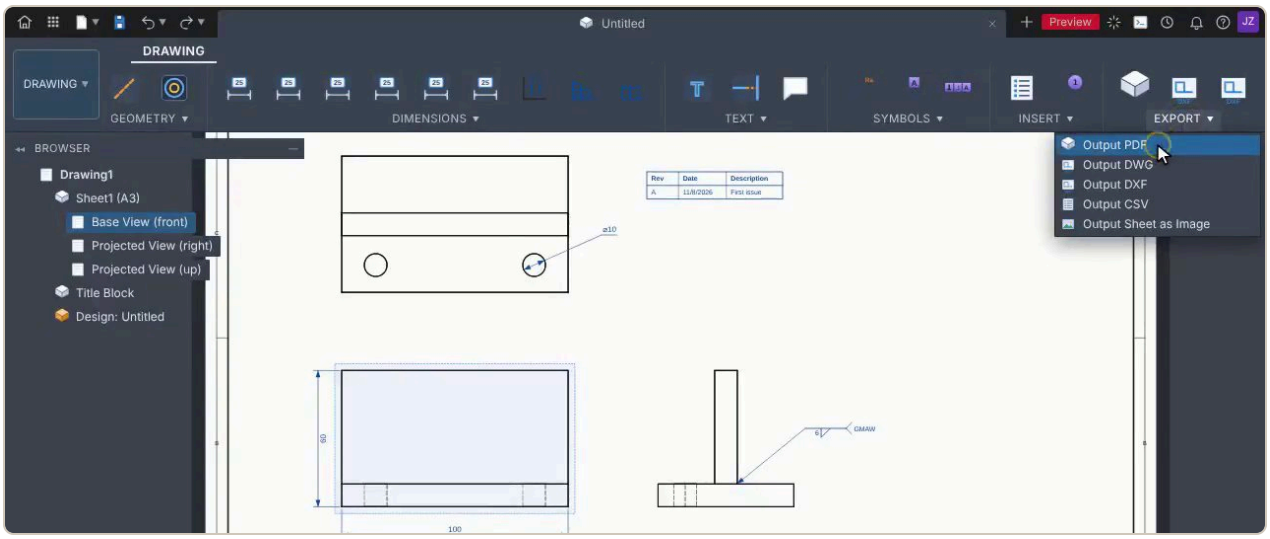
Make a 2D drawing

Turn a part into a 2D drawing: place a front view and projected views, add dimensions, a weld symbol and a revision table, then save it as a PDF.

In this lesson you'll turn a 3D part into a flat 2D drawing, the kind you can print and hand to a shop.

1. Here's a simple bracket: a base plate with two holes, and an upright plate on top.

2. Click Design at the top left, point to Drawing, and choose From Design.

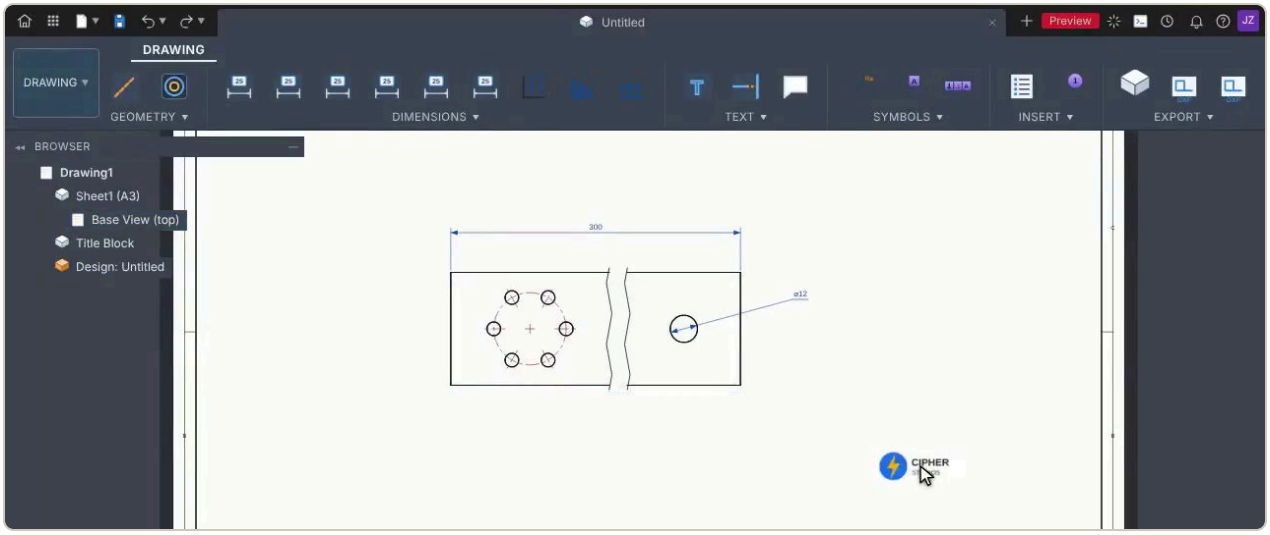


3. The Create Drawing window sets the standard, units and sheet size. The defaults are fine, so click OK.
4. A blank sheet opens, with the view set to Front. Click on the sheet to place it, then click OK.
5. Now click Projected View, and click the front view. Move to the right and click, to add the side view. Move up and click, to add the top view.
6. Scroll the mouse wheel to zoom in on the views.
7. Time for dimensions. Click Dimension, click the two bottom corners of the front view, then click below to place it. 100 mm.
8. Do the same up the left side for the height, 60 mm.
9. Click a hole in the top view, and place it. That's a diameter of ten.
10. Now a weld symbol. Open the Symbols menu and choose Weld Symbol.
11. It's a fillet weld, 6 mm, on the arrow side. Type G M A W as the tail note, the welding process, and click Place.
12. Click the joint where the upright meets the base, then click again to place the symbol.
13. Tables work too. Open the Insert menu and choose Table. You type one row per line. Click Place.
14. Click on the sheet where the table's top left corner goes.
15. Last, open the Export menu and choose Output PDF. Ironcast saves the whole sheet as a PDF file.

That's it. Views, dimensions, symbols and a table, ready to print or send.

Drawing extras: center marks, breaks and images

Add a Center Mark Pattern to a bolt circle, shorten a long view with Break View, dimension across the break, and insert a logo image.



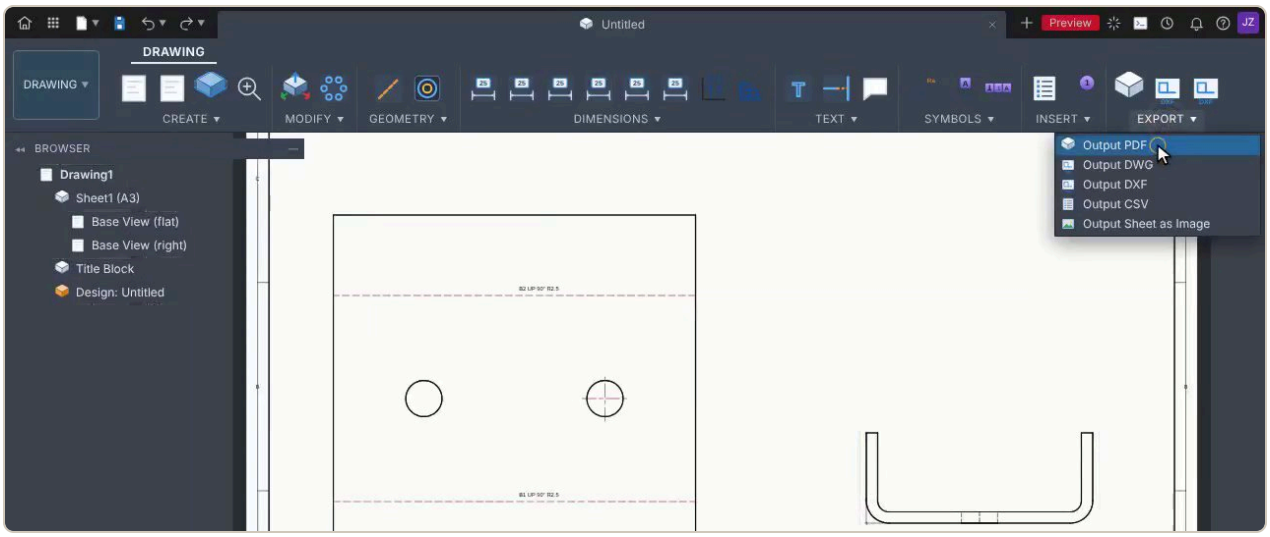
In this lesson you'll add some finishing touches to a drawing: center marks, a break view, and a picture.

1. Here's a long flat bar, 300 mm, with six small holes on a bolt circle, and one big hole at the other end.
2. Click Design at the top left, point to Drawing, and choose From Design. The defaults are fine, so click OK.
3. The holes go through the top, so set Orientation to Top. Click on the sheet to place the view, then click OK.
4. Scroll the mouse wheel to zoom in on the view.
5. Open the Geometry menu, and choose Center Mark Pattern.
6. Click just one of the small holes. All six get center marks, joined by a bolt circle.
7. The middle of this bar is plain, so let's shorten it. Open the Create menu and choose Break View.
8. Click inside the view where the break starts, then where it ends. The middle is cut out, and the view gets shorter.
9. Now a dimension across the break. Click Dimension, click the top left corner, then the top right corner, and click above to place it.
10. It still reads 300, the real length of the bar, even though the view is shorter.
11. Click the big hole, and place its diameter. 12 mm. Press Escape when you're done.
12. Last, a logo. Slide the toolbar over, open the Insert menu, and choose Image.
13. Click where the picture goes, then pick an image file. Here it's a company logo.
14. It's placed 40 mm wide. Drag it to move it, and right-click it to change its width.

That's it. Center marks, a break view and a picture make your drawings clear and professional.

Drawing a flat pattern

Put a sheet metal part's Flat Pattern on a drawing, label its bends with Bend Identifier, add Edge Extensions at a bend, clean up a centre mark with the Delete key, and output a PDF.



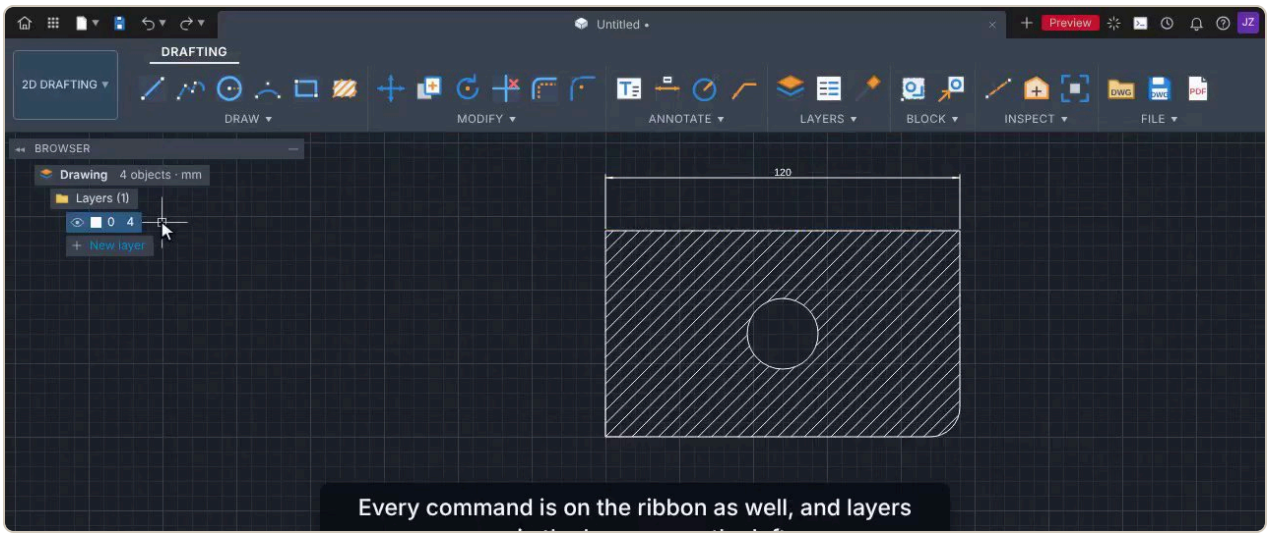
In this lesson you'll put a sheet metal flat pattern on a drawing, and label its bends.

1. Here's a sheet metal U channel, folded from an 80 × 50 sheet, with two holes in the floor.
2. Click Design at the top left, point to Drawing, and choose From Design.
3. Set Sheet Scale to two to one, and click OK.
4. In the view settings, set Orientation to Flat Pattern. Click on the sheet to place it, then click OK.
5. There's the part, unfolded flat. Now a side view, to see the bends. Open the Create menu and choose Base View.
6. Set Orientation to Right, click to the right of the flat pattern, and click OK.
7. Roll the mouse wheel to zoom in. The dashed lines across the flat pattern show where it bends.
8. Open the Insert menu, choose Bend Identifier, and click the flat view.
9. Zoom in to read one. Each bend gets a label: its number, up or down, 90°, and the inside radius.
10. Let's add centre marks to the holes. Open Geometry, choose Center Mark, and click each hole. Press Escape when you're done.
11. Changed your mind about one? Click the centre mark to select it, and press the Delete key. It's gone.
12. Now the side view. Its corners are rounded by the bends. Open Geometry and choose Edge Extension.
13. Click the straight edge on one side of the bend, then the straight edge on the other side.
14. Zoom in on the corner. Thin lines carry both edges on to the sharp corner where they would meet.
15. When the sheet is ready, open the Export menu and choose Output PDF.

That's it. Flat pattern view, bend labels, edge extensions and a PDF, ready for the shop.

2D Drafting and DWG files

Draw a plate with typed commands the way you would in AutoCAD: rectangle, circle, fillet, hatch and a dimension. Then save it as a real DWG file.



Ironcast can make flat 2D drawings too, and save them as real DWG files.

1. Open the workspace menu at the top left and pick 2D Drafting.
2. You draw by typing short commands, just like AutoCAD. Type R E C and press Enter for a rectangle.
3. Type zero comma zero for the first corner, then 21 comma 70 for the other.
4. Now a hole. Type C for circle, give the center, 60 comma 35, and a radius of 12.
5. Round a corner with Fillet. Type F, then R to set the radius to ten.
6. Click the two sides that meet at the corner, and it is rounded.
7. Type H for hatch and click inside the plate to fill it with section lines. The hole is left open.
8. For a size, type D L I. Click one corner, then the other. The green square shows it snapped to the corner.
9. Then click where the dimension should sit. It measures 120.
10. Every command is on the ribbon as well, and layers are in the browser on the left.
11. To share it, click Save As DWG in the File group, choose the AutoCAD version, and click Save.
12. The file opens in AutoCAD and other CAD programs. Open brings a DWG or DXF back in, and Send to a Sketch turns the drawing into a 3D sketch.

CHAPTER 10

Simulation

Find out how a part behaves before you make it: stress, bending, and where material can come off.

Reading simulation results
Solve a static stress study on a cantilever bar, then explore the Results tab: Stress, Show Min/Max, Probe, Slice Plane, Strain, Displacement and Animate.

Simulation constraints and Clone Study
Hold a bar with Fixed, Frictionless and Prescribed Displacement constraints, solve the static stress study, then clone it, switch the material and compare.

Simulation: degrees of freedom and load cases
Use Degrees of Freedom View to see what a study still lets move, fix it with a constraint, then copy the study with Load Case, suppress its loads, add a new one, simplify the part and solve.

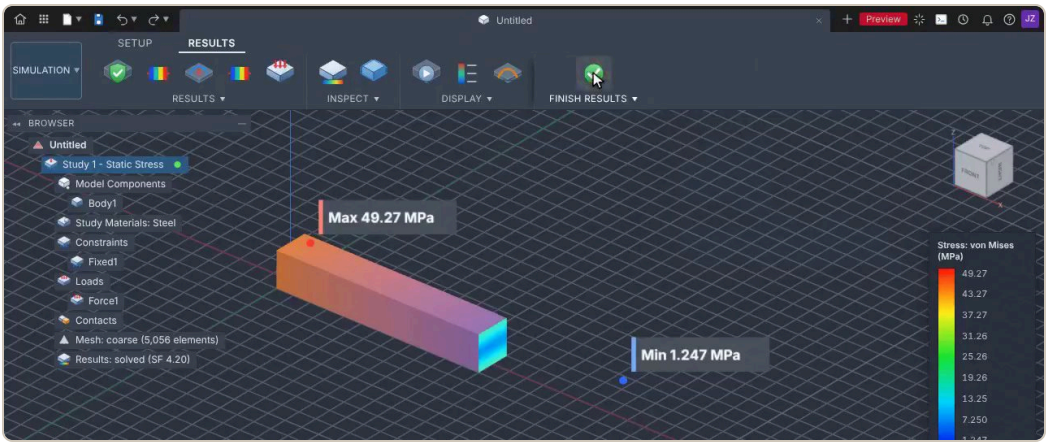
Simulation: Shape Optimization
Let Ironcast show where a part can lose material: start a Shape Optimization study, hold one end, push on the top, solve, and turn the lighter shape into a mesh body.

LESSON 47

Video in Ironcast: Home › Learn › “Reading simulation results” (1:56)

Reading simulation results

Solve a static stress study on a cantilever bar, then explore the Results tab: Stress, Show Min/Max, Probe, Slice Plane, Strain, Displacement and Animate.



In this lesson you'll solve a stress study, then explore the results.

1. Here's a steel bar. The left end is fixed, and a 100 newton force pushes down on the right end.

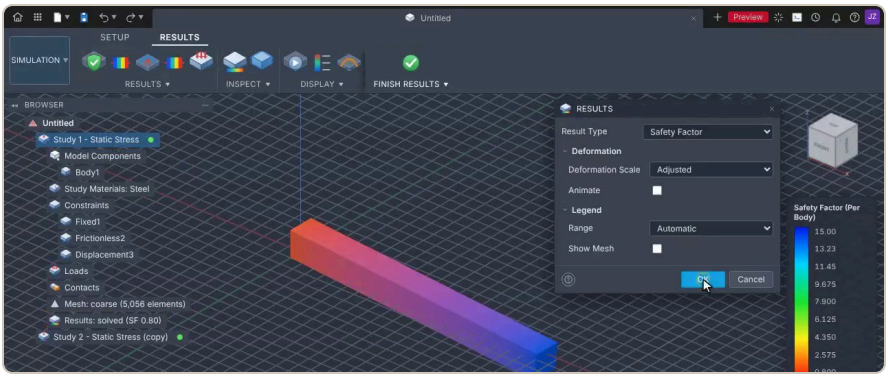
2. Click Solve, on the right of the toolbar. Keep the mesh on Coarse for a quick result, and click OK.
3. Ironcast solves it on this computer and opens the Results tab.
4. It starts with Safety Factor. Click Stress to see von Mises stress. Red is the highest, right where the bar is held.
5. Here you can pick other stress components. Click OK.
6. Open the Inspect menu and choose Show Min/Max. Labels mark the highest and lowest values.
7. Click Probe, then click points on the bar. Each probe shows the stress at that spot.
8. Click Probe again to stop. Then click Strain, to see how much the steel stretches, and OK.
9. Click Displacement, and OK. The free end moves the most, about two tenths of a millimeter.
10. The real bend is tiny, so click Deformation Scale and choose 100 times.
11. Now click Animate, and watch the bar bend up and down.
12. Click Animate again to stop. Go back to Stress, click OK, and choose Clear Probes from the Inspect menu.
13. Now click Slice Plane. Set Direction to X, and Position to 25%.
14. The cut shows the inside. It's red at the top and bottom, and blue in the middle.
15. Change Position to 60, and the slice moves along the bar. Click OK to keep it.
16. When you're done, click Finish Results to go back to the study setup.

Now you can read stress, strain and displacement, and look inside with probes and slices.

LESSON 48 Video in Ironcast: Home › Learn › “Simulation constraints and Clone Study” (1:48)

Simulation constraints and Clone Study

Hold a bar with Fixed, Frictionless and Prescribed Displacement constraints, solve the static stress study, then clone it, switch the material and compare.



In this lesson you'll hold a part with three kinds of constraints, then clone the study to compare materials.

1. Here's a steel bar in a new static stress study, with no constraints yet.

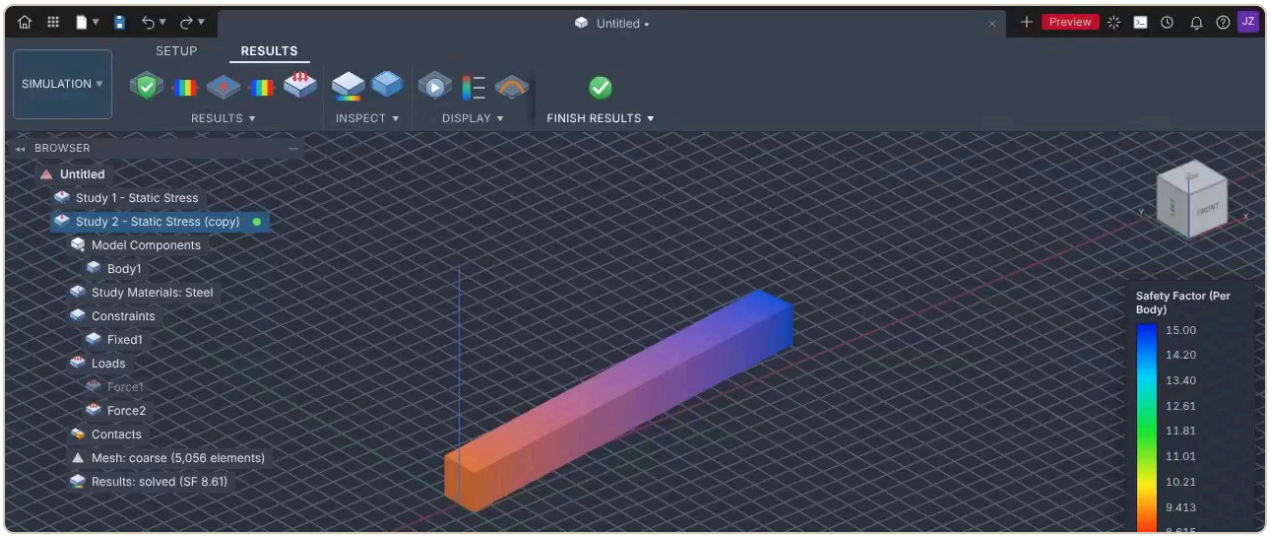
2. Click Structural Constraints. The type is Fixed. Click the left end of the bar, and click OK. That end can't move at all.
3. Click Constraints again, and change Type to Frictionless. Click the long front face, and OK.
4. Frictionless lets the bar slide along that face, so it can bend up and down, but not sideways.
5. Turn the view so you can see the other end of the bar.
6. Open Constraints once more, and pick Prescribed Displacement. Then click the right end face.
7. Uncheck U X and U Y, type minus one for Z Displacement, and click OK. That end gets pushed down 1 mm.
8. Now click Solve, on the right of the toolbar, and click OK.
9. The lowest safety factor is 0.8. Below one, the steel would bend for good.
10. To try another material, click Finish Results, then Study, Clone Study.
11. A copy appears with all three constraints. Click Study Materials, choose Override, and pick Aluminum.
12. Click OK, then Solve and OK again.
13. Same bend, but the safety factor is now above three. Aluminum is less stiff, so it has much less stress.
14. Click Study 1, then Safety Factor and OK, to flip back to the steel result.

That's it. Hold parts the way they're really held, and clone studies to compare ideas.

LESSON 49 Video in Ironcast: Home › Learn › “Simulation: degrees of freedom and load cases” (1:31)

Simulation: degrees of freedom and load cases

Use Degrees of Freedom View to see what a study still lets move, fix it with a constraint, then copy the study with Load Case, suppress its loads, add a new one, simplify the part and solve.



In this lesson you'll check degrees of freedom, then try a second load case on the same part.

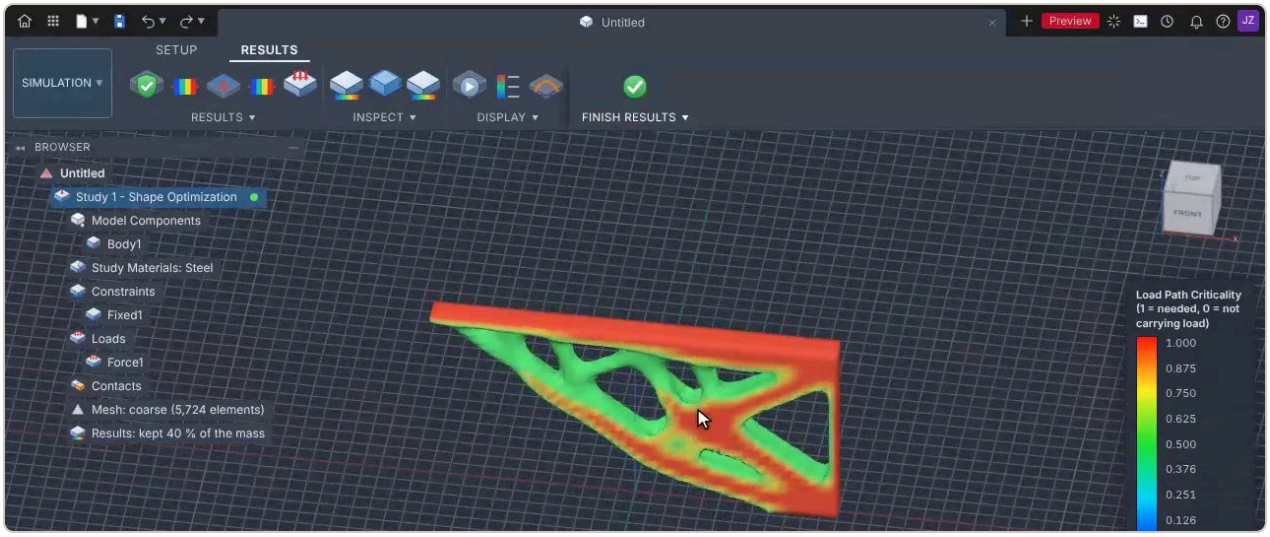
1. Here's a steel bar with a small hole, in a new static stress study. Nothing holds it yet.

2. Open Display, and choose Degrees of Freedom View. Six are free: the bar could slide or spin any way.
3. A solve would fail like this. Close it, click Structural Constraints, click the left end, and OK.
4. Run Degrees of Freedom View again. Now it's zero, so the study can be solved.
5. Click Structural Loads, and click the top face. A 100 newton force pushes down. Click OK.
6. To try a different load, open Loads, Load Case, and choose New Load Case. A copy of the study appears.
7. In the copy, open Load Case again and choose Suppress All Loads. The downward force turns off.
8. Now add a sideways load. Click Structural Loads, click the front face, and OK.
9. The small hole hardly matters here. Open Simplify, Remove Features, click the bar, and OK. The hole fills in.
10. Click Solve, on the right of the toolbar, and click OK.
11. The bar now bends sideways. The lowest safety factor, in orange, is at the fixed end.
12. Study 1 still has the downward load, so you can switch between the two cases and compare.

LESSON 50 Video in Ironcast: Home › Learn › “Simulation: Shape Optimization” (1:21)

Simulation: Shape Optimization

Let Ironcast show where a part can lose material: start a Shape Optimization study, hold one end, push on the top, solve, and turn the lighter shape into a mesh body.



Shape Optimization shows you where a part can lose weight and still do its job.

1. Here is a solid steel plate. It will be held at one end, with a load pushing down on top, like a shelf bracket.
2. Open the workspace menu and pick Simulation. The New Study box opens.
3. Choose Shape Optimization as the study type. Electronics Cooling and Injection Molding are new in this list too. Click OK.
4. First hold the part. Click Constraints, click the end face on the right, and click OK.

5. Now the load. Click Loads, click the top face, and type 500 newtons.
6. In Study Settings, Mass to keep is the target. 40% means more than half of the plate comes off.
7. Click Solve, keep the mesh on Coarse, and click OK. Ironcast works it out on this computer.
8. When it finishes, the Results tab opens. The browser shows it kept 40% of the mass.
9. What is left is where the load really travels. Red carries the most, and the shape looks like a truss.
10. Open the Inspect menu and choose Promote Shape to Mesh Body. The new shape is now a body in your design, and the solid plate is hidden.

CHAPTER 11

Machining (Manufacture)

Toolpaths for a mill or router, a preview of the cut, and G-code for your machine.

Manufacture: 2D milling
Mill a plate with a pocket, an outside profile and holes: Setup and stock, Face, 2D Pocket, 2D Contour, Drill and 2D Chamfer, then Generate, Simulate, Setup Sheet and Post Process.

Manufacture: 3D toolpaths
Make a milling setup for a filleted block, add 3D Contour, Radial and Spiral finishing and a 2D Circular pass in a round pocket, then Generate, Simulate and Post.

Manufacture: finishing toolpaths
Finish a filleted block with a blind pocket using Steep and Shallow, Scallop, Pencil, Horizontal, Ramp and Project, then Generate and Simulate.

Manufacture: Morph, Flow and Manage
Add Morph, Morphed Spiral, Flow and Slot toolpaths to a filleted block, then pick a machine in Machine Library, set a default post and rebuild everything in Task Manager.

LESSON 51 — Video in Ironcast: Home › Learn › “Manufacture: 2D milling” (1:56)

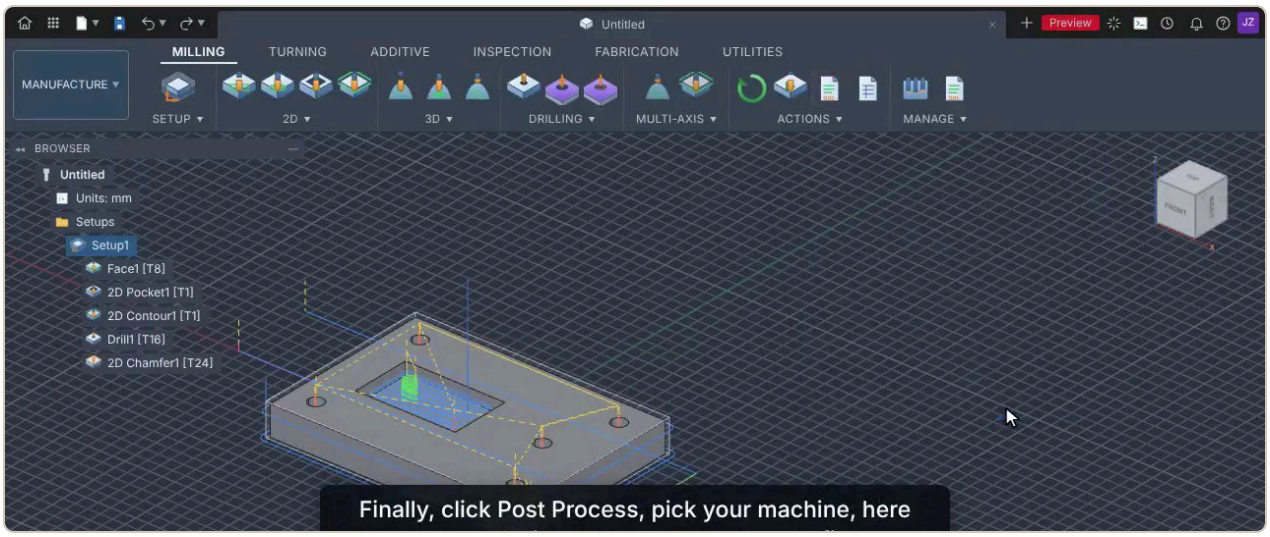
Manufacture: 2D milling

Mill a plate with a pocket, an outside profile and holes: Setup and stock, Face, 2D Pocket, 2D Contour, Drill and 2D Chamfer, then Generate, Simulate, Setup Sheet and Post Process.

Let's mill a plate with 2D toolpaths.

1. Here's a 12 mm plate with a pocket on top and 11 mm holes.

2. Click Design at the top left and switch to Manufacture. The Setup dialog opens, with the origin at the top front left of the stock.

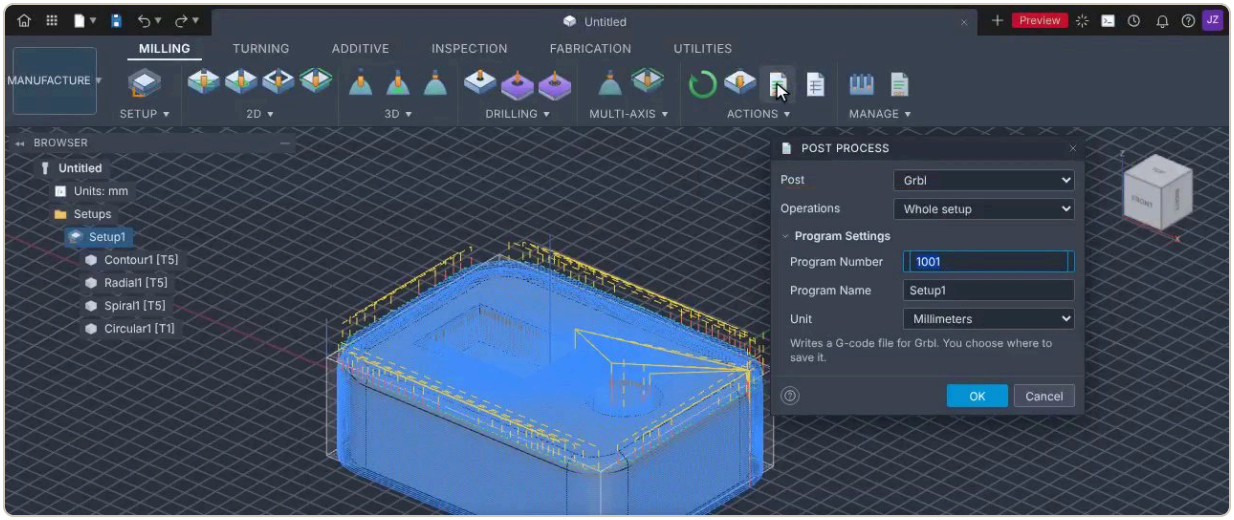


3. Open the Stock tab. The stock is a box 1 mm bigger on the sides and top. Click OK.
4. Open the 2D menu and choose Face. A 50 mm face mill skims the top of the stock flat. Click OK.
5. Next, 2D Pocket. It finds the pocket by itself and clears it with a 6 mm end mill. Click OK.
6. Now 2D Contour. It cuts around the outside of the plate, all the way to the bottom, to free the part.
7. For the holes, open Drilling and choose Drill. On the Geometry tab, it picks every 6 mm hole. Click OK.
8. Last, 2D Chamfer. On the Passes tab, set Chamfer Width to one and a half, and click OK. It breaks the sharp top edges.
9. Click Setup1 in the browser, then Generate, to build all five toolpaths.
10. Click Simulate and choose Whole setup. The tool runs every pass and cuts the stock, so you can check it before machining.
11. Let it run a little longer, then click OK to close the simulation.
12. Open Actions and choose Setup Sheet. Ironcast saves a page for the shop, listing the stock, and each operation's tool and run time.
13. Finally, click Post Process, pick your machine, here Grbl, and click OK to save the G-code file.

That's it. Set up the stock, add your 2D toolpaths, then generate, simulate, and post.

Manufacture: 3D toolpaths

Make a milling setup for a filleted block, add 3D Contour, Radial and Spiral finishing and a 2D Circular pass in a round pocket, then Generate, Simulate and Post.



In this lesson you'll make 3D finishing toolpaths for a CNC mill: Contour, Radial, Spiral, and a Circular pass for a round pocket.

1. Here's a block with rounded edges, a square pocket, and a round pocket.
2. Click the Design button at the top left, and switch to the Manufacture workspace.
3. Click Setup. The origin goes at the top front left of the stock. Click OK, and a stock box appears around the part.
4. Open the 3D menu and choose Contour. It uses the 6 mm flat end mill. Click OK.
5. Contour cuts around the walls, one level at a time, all the way down.
6. Now choose Radial from the 3D menu. It picks a ball end mill. Click OK. The passes run out from the center, like spokes on a wheel.
7. Next, open 3D and choose Spiral, and click OK. One long spiral sweeps over the top, following every curve.
8. For the round pocket, open the 2D menu and choose Circular. On the Geometry tab, it finds every hole bigger than the tool.
9. Click OK. The tool spirals down inside the round pocket.
10. Click Setup1 in the browser, then click Generate, to build all four toolpaths together.
11. Click Simulate. The tool plays through every move and cuts away the stock, so you can check it before you run the real machine.
12. Let it run a little longer, then click OK to close the simulation.

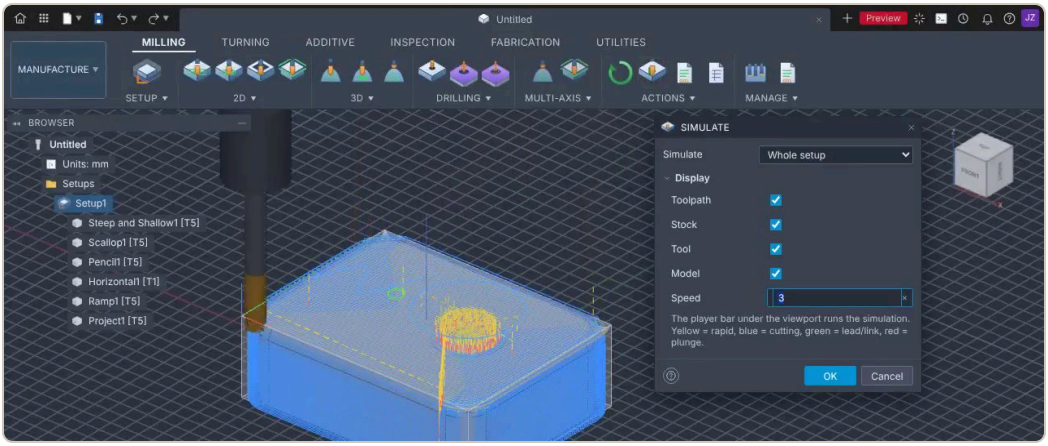
13. Finally, click Post Process. Pick your machine, here Grbl, and click OK. Ironcast saves the G-code file for your mill.

That's it. Set up the stock, add your toolpaths, then generate, simulate, and post.

LESSON 53 Video in Ironcast: Home › Learn › “Manufacture: finishing toolpaths” (1:36)

Manufacture: finishing toolpaths

Finish a filleted block with a blind pocket using Steep and Shallow, Scallop, Pencil, Horizontal, Ramp and Project, then Generate and Simulate.



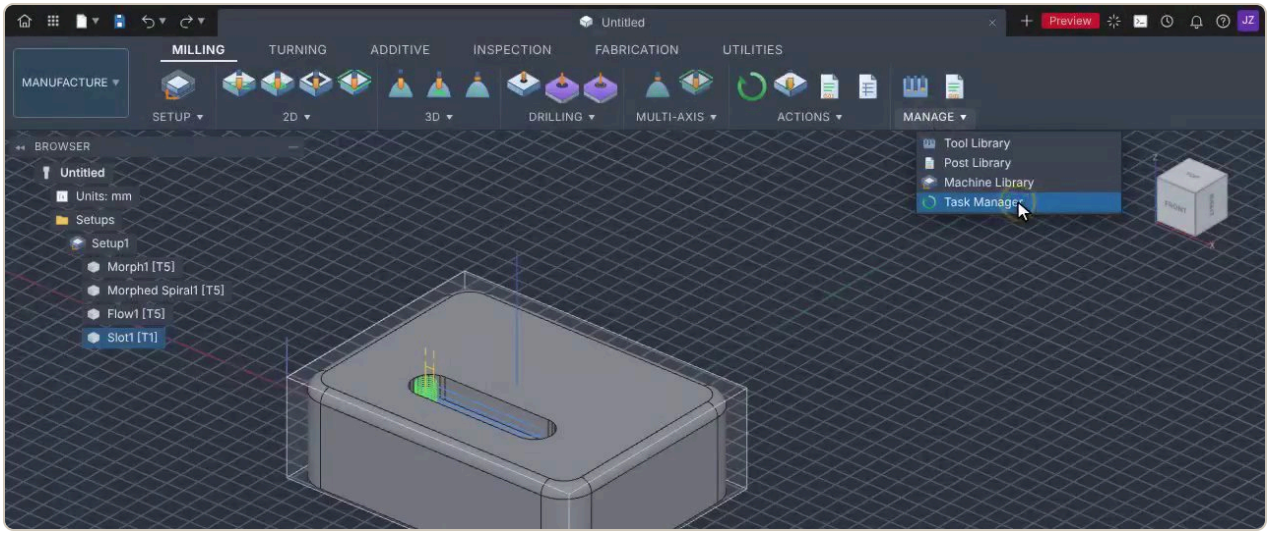
In this lesson you'll add 3D finishing toolpaths for a smooth milled surface.

1. Here's a block with rounded edges and an 8 mm deep pocket. Above it floats a sketch.
2. Click Design at the top left, and switch to Manufacture. In the Setup dialog, click OK to put a stock box around the part.
3. Open the 3D menu and choose Steep and Shallow. On the Passes tab, set Threshold Angle to 40°.
4. Steeper walls get level-by-level passes, and flatter areas get back-and-forth passes. Click OK.
5. Next, 3D, Scallop, and OK. Its passes step in from the outline with an even gap.
6. Now Pencil. It only cuts tight inside corners, like the bottom edge of the pocket.
7. Horizontal finishes only the flat areas: the top, and the pocket floor.
8. Ramp works on the walls, sloping down steadily as it goes around.
9. Last, choose Project. On the Passes tab, pick the Engrave sketch, and a depth of 0.2.
10. Click OK. The sketch lines drop onto the part, and the tool follows them to engrave.
11. Click Setup1, then Generate, to build all six toolpaths.
12. Click Simulate, and set it to Whole setup. The tool runs every pass so you can check it before machining.

That's it. Each finishing toolpath handles a different part of the shape.

Manufacture: Morph, Flow and Manage

Add Morph, Morphed Spiral, Flow and Slot toolpaths to a filleted block, then pick a machine in Machine Library, set a default post and rebuild everything in Task Manager.



In this lesson you'll try Morph, Morphed Spiral, Flow and Slot, then the Manage tools.

1. Here's a block with rounded corners and a slot cut into the top.
2. Click Design at the top left, and switch to Manufacture. In the Setup dialog, click OK to put a stock box around the part.
3. Open the 3D menu and choose Morph. It picks a 6 mm ball end mill. Click OK.
4. Morph passes are rings shaped like the part's outline, shrinking in toward the middle.
5. Next, 3D, Morphed Spiral, and OK. This is one continuous cut that spirals in from the outside.
6. Now 3D, Flow, and OK. Flow lines run across the part, spreading evenly from the front edge to the back.
7. For the slot, open the 2D menu and choose Slot. Click OK, and it clears the slot 2 mm at a time.
8. Now the Manage tools. Open Manage, Machine Library. Each machine lists its travel, top speed and post.
9. Click Use on the Haas Mini Mill. Its top speed is 6000 r p m, so Ironcast caps all four spindle speeds.
10. Next, Manage, Post Library. The Haas post came with the machine. Click Set Default on another post to switch.
11. Close it, and open Manage, Task Manager. It lists every toolpath with its status and machining time.
12. Click Generate All to rebuild every toolpath at once. The total time shows at the bottom.

That's it. Pick your machine and post once, and keep every toolpath up to date from one place.

CHAPTER 12

Electrical: control panels and circuit boards

Ladder diagrams with a PLC simulator, motor circuits and panel layout, and circuit boards.

Ladder logic basics
Start a Control Panel project, draw a motor start/stop rung with a seal-in contact, label the buttons, add a rung comment and switch to the NEMA wiring look.

Run the PLC simulator
Turn on the PLC simulator, press START and STOP, watch the rungs energize and a timer count, then force inputs from the tag window.

Motors, power sheet and panel layout
Add a motor circuit and let Ironcast fill in the full-load amps from the horsepower, see the power sheet, then the to-scale panel layout, the parts catalog and the bill of materials.

Panel heat load, cable schedule and parts
Check how hot a control panel will get and how big a fan it needs, size motor feeder cables for voltage drop, price parts in the catalog and bill of materials, and export a tag list.

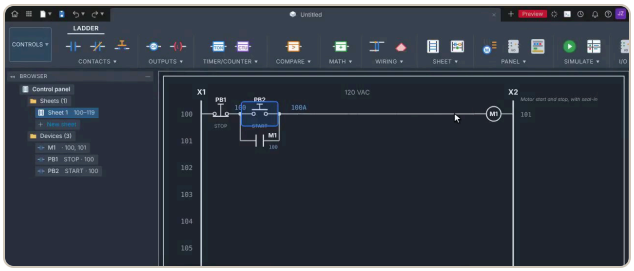
Electronics: schematic to PCB
Open the Electronics workspace, place a resistor, LED and battery on the schematic, wire them, run ERC, lay out and route the 2D PCB, run DRC, and push it to a 3D board.

LESSON 55

Video in Ironcast: Home › Learn › “Ladder logic basics” (1:45)

Ladder logic basics

Start a Control Panel project, draw a motor start/stop rung with a seal-in contact, label the buttons, add a rung comment and switch to the NEMA wiring look.



In this lesson you'll draw your first ladder diagram: a motor start and stop circuit.

1. On the home screen, click New, pick Control Panel, and click Create New.
2. This is a ladder sheet. Power flows from the left rail to the right, along lines called rungs.
3. Open the Contacts menu, pick Stop Pushbutton, and click at the start of the first rung.

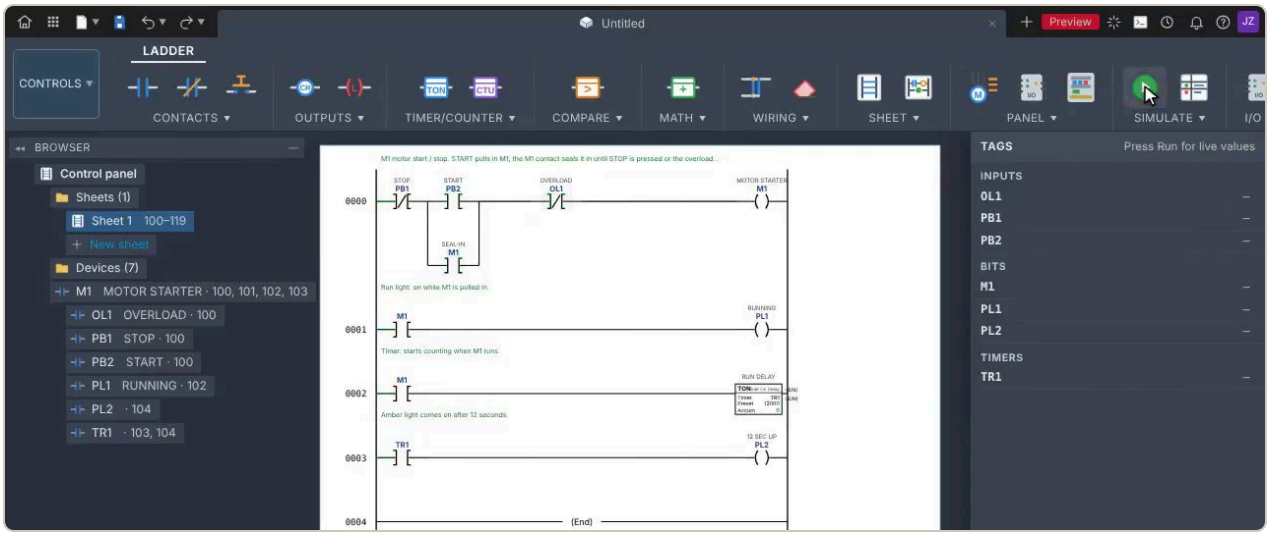
4. Next, click Start Pushbutton, and place it right after the stop button.
5. From the Outputs menu, pick Starter Coil, and click on the same rung. Outputs go next to the right rail.
6. The coil is named M1. To keep it running after you let go of START, click NO Contact, and place it below START. Ironcast names it M1 too.
7. Now wire it alongside START. Click Branch Wire, then click the gap on each side of the new contact.
8. That's a seal-in. Once START is pressed, the M1 contact keeps the coil on until STOP is pressed.
9. Let's label the buttons. Double-click the stop button, type STOP in Description, and click OK.
10. Do the same for the start button.
11. To explain a rung, right-click it, choose Rung comment, type a note, and click OK.
12. This is the PLC ladder look, in the Allen-Bradley style. For a wiring diagram, open the Sheet menu and choose Switch Ladder Style.
13. Now it's the NEMA look, with standard symbols, the control voltage on top, and wire numbers. The same command switches back.

That's a complete start and stop circuit. Next, you'll run it in the PLC simulator.

LESSON 56 Video in Ironcast: Home › Learn › “Run the PLC simulator” (1:27)

Run the PLC simulator

Turn on the PLC simulator, press START and STOP, watch the rungs energize and a timer count, then force inputs from the tag window.



In this lesson you'll test a ladder circuit with the built-in PLC simulator.

1. Here's the start and stop circuit from the last lesson, plus a timer rung. It waits 12 seconds, then turns on an amber light.
2. To test it, click Run in the Simulate group. Green shows where power can flow right now.

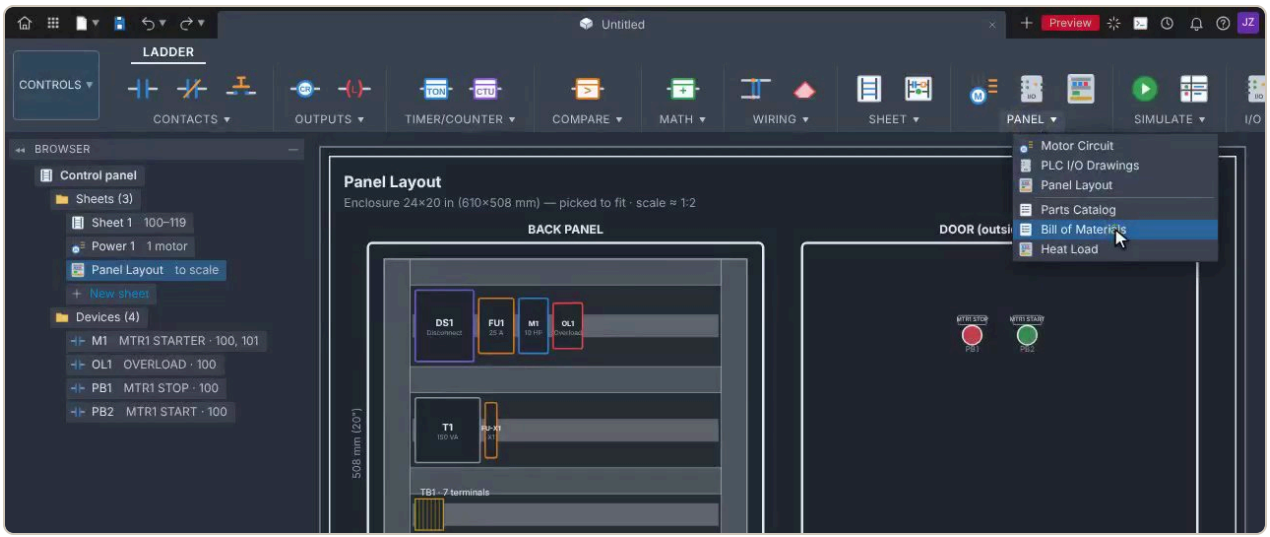
3. Click and hold START. The M1 coil turns on, the seal-in contact holds it on, and the green run light lights up.
4. The timer starts counting too. Watch its value climb toward 12000 milliseconds.
5. When time is up, the timer contact closes, and the amber light comes on.
6. Now press STOP. Everything drops out, and the timer resets.
7. Click Tag Window to see every tag with its live value: the inputs, the on and off bits, and the timer.
8. You can force an input from here without touching the drawing. Click Force next to PB2, the start button, and the motor starts.
9. Now force PB1, the stop button. The motor stops, just as if someone were holding STOP.
10. Forced inputs show in orange, so you won't forget them.
11. When you're done, click Run again to turn the simulator off.

Now you can test your logic before you wire a real panel. Next, you'll add a motor and lay out the panel.

LESSON 57 Video in Ironcast: Home › Learn › “Motors, power sheet and panel layout” (1:47)

Motors, power sheet and panel layout

Add a motor circuit and let Ironcast fill in the full-load amps from the horsepower, see the power sheet, then the to-scale panel layout, the parts catalog and the bill of materials.



In this lesson you'll add a motor, and Ironcast will draw its power wiring, lay out the panel and list the parts.

1. Start with a new Control Panel project. In the Panel group, click Motor Circuit.
2. The New Motor Circuit box opens. Give the motor a description, like CONVEYOR.
3. Now set the horsepower to ten. Ironcast fills in the full-load amps for you: 14 amps for a ten horsepower motor at 460 volts.
4. It also suggests the fuse and wire sizes. Leave Draw its control rung ticked, scroll down, and click OK.

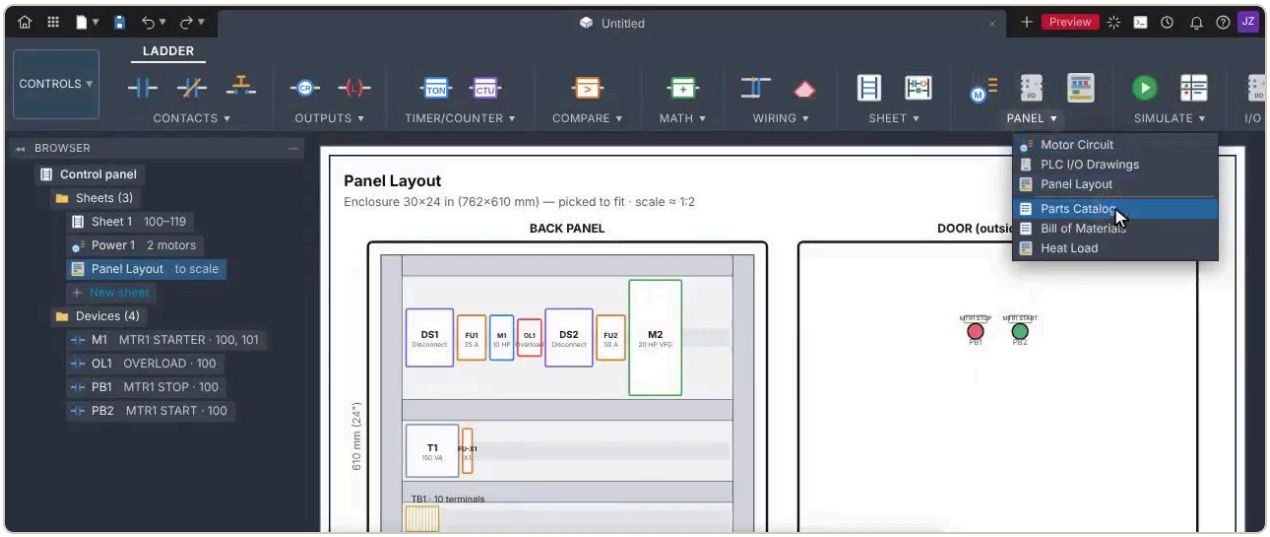
5. This is the power sheet. Three-phase power comes in at the top, through the disconnect, the fuses, the starter contacts and the overload, down to the motor.
6. Ironcast also drew the motor's start and stop rung. Click Sheet 1 in the browser to see it.
7. Now click Panel Layout. Ironcast draws the panel to scale, in the smallest standard enclosure that fits.
8. The back panel holds the disconnect, fuses, starter and terminals on DIN rails. The door shows the pushbuttons.
9. To choose real parts, open the Panel menu and pick Parts Catalog. It comes with starter parts and their sizes. Check the ones marked verify with your supplier.
10. Click Done. Then open the Panel menu again and pick Bill of Materials.
11. On the left, pick a part for each device. For the M1 starter, choose the 18 amp contactor, and its part number fills into the list.

That's a motor, its power sheet, a panel layout and a parts list, all from one project.

LESSON 58 Video in Ironcast: Home › Learn › “Panel heat load, cable schedule and parts” (1:55)

Panel heat load, cable schedule and parts

Check how hot a control panel will get and how big a fan it needs, size motor feeder cables for voltage drop, price parts in the catalog and bill of materials, and export a tag list.



In this lesson you'll check a control panel for heat, size the motor cables, and price the parts.

1. Here's a control panel with two motors: a ten horsepower conveyor, and a 20 horsepower pump on a drive.
2. The heat check needs a box size, so open the Panel menu and pick Panel Layout.
3. Then open the Panel menu again, and choose Heat Load.
4. The parts make about 470 watts of heat, but the box can only shed about 56. Too hot without cooling.
5. Ironcast suggests a fan of about 73 CFM. The drive makes almost all of the heat.

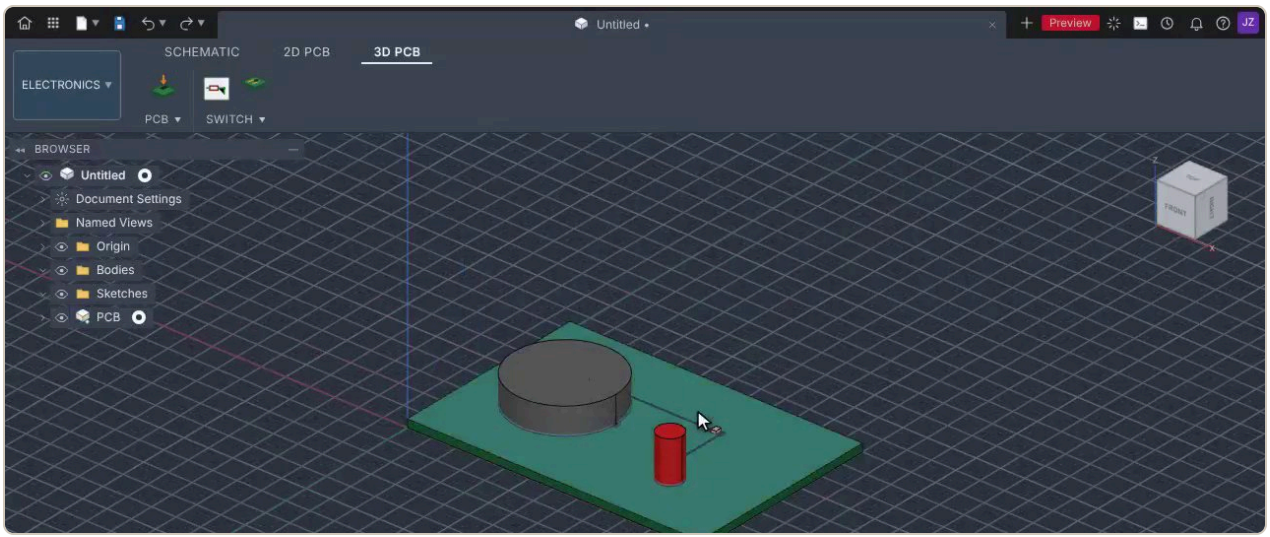
6. Try a cooler room. Set the outside air to 80°, and click Update.
7. Click Close. Next, open the Output menu on the far right, and choose Cable Schedule.
8. Type each cable run: 300 feet for the conveyor, and 400 for the pump.
9. Click Update. To keep voltage drop under three percent, both cables get bigger, and the note says what they were upsized from.
10. Close it, and open Parts Catalog from the Panel menu. Here you enter real prices, and real watts for the heat check.
11. Give the 18 amp contactor a price of 85 dollars, and the 12 to 18 amp overload 60.
12. Click Done, and open Bill of Materials. Pick the contactor for M1, and the overload for OL1.
13. Now the list has part numbers, prices and a total.
14. To share your tags, open the Output menu, choose Export, and click Tag list. A CSV file is saved.

That's heat load, cable sizes and a priced parts list. Always check results with the code and your supplier.

LESSON 59 Video in Ironcast: Home › Learn › “Electronics: schematic to PCB” (1:35)

Electronics: schematic to PCB

Open the Electronics workspace, place a resistor, LED and battery on the schematic, wire them, run ERC, lay out and route the 2D PCB, run DRC, and push it to a 3D board.



Let's design a small circuit board.

1. Click Design at the top left and switch to Electronics. It opens on the Schematic tab, with a blank sheet.
2. Click Place Component. The library sorts parts by type. Search for resistor, and double-click it.
3. Click on the sheet to drop it, then press Escape.
4. Do the same for an LED, and for a battery.
5. Now click Net. Click a pin, then another pin, to draw a wire between them.

6. Wire the battery plus to the resistor, and the battery minus to the LED. The wires bend around the corner for you.
7. Click ERC, the electrical rules check. No problems found, so click Close.
8. Click Switch to 2D PCB. The parts wait beside the board, and yellow airwires show what must connect.
9. Drag each part onto the board: the battery holder, the LED, and the resistor.
10. Click Route. Click a pad, then the matching pad, to draw a copper trace.
11. For the rest, click Autorouter and OK. It routes every airwire that's left.
12. Click DRC, the design rules check, to look for clearance problems. Then click Close.
13. Finally, click Push to 3D PCB. Ironcast builds the board in 3D, with the battery holder, the LED and the resistor.

That's it. Draw the schematic, lay out the board, check it, and see it in 3D.

MORE TOOLS

Generative Design

Instead of drawing the shape yourself, you tell Ironcast what the part has to do and it works out shapes that do it.

1. Model the pieces that must exist: where the part bolts on, and where the load goes in. These are the **preserve** bodies. If something must stay clear (a shaft, a bolt head, room for a hand), model that space as a body too. Those are **obstacles**.
2. Click the workspace button and choose **Generative Design**.
3. Click **Preserve Geometry** and pick the bodies that must stay. They turn green. Click **Obstacle Geometry** and pick the bodies to keep clear of. They turn red.
4. Add supports with **Constraints** and forces with **Loads**, the same way as in a stress study. Put them on preserve bodies.
5. Open **Generative Setup**. Choose the goal (the lightest shape that reaches a safety factor, or the stiffest shape at a set mass), the manufacturing methods to try, and the materials.
6. Click **Generate**. Ironcast fills the space around the preserve bodies and removes material the loads do not need, once for each manufacturing method. Press Esc to stop early.
7. The **Explore** tab lists the outcomes with mass, safety factor, stress and how far each one bends. Click a row to see its shape.
8. Click **Create Body** to put the outcome you like into your design.

Unrestricted	Any shape the loads ask for.

Additive (3D printing)	Prints standing up (Z up) without supports: no overhang flatter than about 45°.
3-axis milling	Can be cut from the top: no material hanging over empty space.
2D cutting	The same outline all the way through, for laser, waterjet or plasma cutting.

> Generating takes a few minutes. Outcomes are not saved with the design, so after reopening a file, click Generate again; your setup is saved. The stress figure leaves out the sharpest half a percent of points, because the small cubes used for the calculation read too high at their corners. Treat every outcome as an engineering estimate.

MORE TOOLS

Other tools worth knowing

Several views at once

On the navigation bar at the bottom, open **Viewports** and choose two, three or four views. Your working view stays at the top left; the others show the model from the top, front and right. Double-click a side view to turn your main view to that side.

Filament Library

In the Simulation workspace, open **Materials** › **Filament Library**. It lists 3D printing filaments by brand. Where the maker publishes strength numbers measured on printed test bars, Ironcast runs a standard stress test with them. **Use** puts a filament on your bodies so mass and stress studies use its real numbers. Filaments without published numbers are listed with no stress test.

Wood, concrete and fabric textures

The appearances Oak, Walnut, Maple, Pine, Concrete, Fabric - Canvas and Sitework - Grass show a texture. **Texture Map Controls** lets you change how the texture is laid on a body, its size, its turn and its position.

Showing and hiding parts of a sketch

Right-click a sketch in the browser to hide its profile shading, show its dimensions, or hide its projected and construction geometry. These choices are saved with the design.

Exploded drawings

Once you have an animation storyboard, click the workspace button and choose **Drawing** › **From Animation**. Ironcast makes a sheet showing the parts where the storyboard puts them, with trail lines,

numbered balloons and a parts list.

Scripts

Ironcast can be driven with Python or JavaScript. Press **Shift+S** to open Scripts and Add-Ins. A short Python script that builds a part looks like this:

```
import ironcast as ic
ic.new_document()
base = ic.sketch('XY').rect(40, 20)
ic.extrude(base, 3, op='new')
ic.fillet(1, 'vertical')
ic.holes([(-12, -2, 3), (12, -2, 3)], 4.5)
```

The full list of script commands is in Ironcast under **?** › Scripting API.

HELP

When something goes wrong

A step in the timeline turns red

That step could not be built, usually because something earlier changed. Hover over it to read why. Double-click it to fix its settings, or press Ctrl+Z to undo the change that broke it. Common causes: a fillet bigger than the edge it sits on, a shell wall thicker than the part, or a sketch that is no longer a closed shape.

A sketch will not extrude

Extrude needs a closed shape, called a profile. A closed profile fills in with color. If yours does not, look for a small gap between two lines, or two lines that cross without meeting at a point.

You are lost in the view

Press **F6** to fit the model on screen, or click the little house by the ViewCube.

Windows blocked a save

Windows “Controlled folder access” can stop programs from writing to Documents. Ironcast tells you when this happens. Save to another folder, or allow Ironcast in Windows Security.

The error log

When something goes wrong, Ironcast writes it down with the steps that led to it. Click **?** › **Error Log** to see the list and where the file is. Nothing in it is sent anywhere. If you report a problem, that file is the most useful thing to include.

Privacy

Your designs stay on your computer. Ironcast does not have an account, does not track you and does not send your files anywhere. It goes online only to check for updates, which sends nothing about you or your work, and to open a supplier's parts catalog when you choose to.

LAST PAGE

About this guide

This guide was written for Ironcast 0.19.1. Ironcast changes often, so a button may have moved since. The video lessons inside Ironcast (Home › Learn) are updated with the program.

Ironcast is made by CipherForgedStudios. It is free to download and use during the beta, and is provided as-is, without warranty. It is built on open-source software, including OpenCascade, three.js, Pyodide and replicad, each used under its own license.