**Agenda for Your Lesson:** Autodesk Fusion 2025

Welcome to your personalized introduction to Autodesk Fusion! This 15-page guide is designed to be a clear, straightforward journey into the world of 3D modeling. Think of it as your personal co-pilot as you learn to navigate this powerful software.

Our goal isn't to make you an expert overnight, but to build a solid and confident foundation. We'll start with the absolute basics—what Fusion is and how to look at it—and steadily build from there. Each page is a self-contained lesson that flows into the next. We'll cover how to draw 2D sketches, turn them into 3D shapes, modify those shapes, and even assemble different parts together. We'll also touch on preparing your model for the real world through manufacturing and creating technical drawings.

This lesson is designed specifically for you. It's broken into small, manageable chunks. At the end of each page, you'll have control. Simply tell me to "proceed" when you're ready for the next piece of information. This way, we can move at a pace that works best for you.

Let's get started.

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**Page 1: What is Autodesk Fusion & The Golden Rule**

**What is Fusion? A Digital Workshop** 🧰

Imagine you have a complete workshop in your computer. You have a drafting table for sketching ideas, a workbench with tools to build those ideas, and even a lab to test if your creation is strong enough. Autodesk Fusion is exactly that—an all-in-one digital workshop.

Professionals use fancy acronyms, but let's break them down simply:

- **CAD (Computer-Aided Design):** This is the **designing** part. It's like a super-powered pencil and paper where you draw 2D sketches and then pull them into 3D shapes, like sculpting digital clay.
- **CAM (Computer-Aided Manufacturing):** This is the **making** part. Once you have a 3D model, CAM helps you create instructions for real-world machines, like 3D printers or CNC mills, to actually build it.
- **CAE (Computer-Aided Engineering):** This is the **testing** part. Before you make anything, CAE lets you run simulations. You can digitally apply pressure to your model to see where it might bend or break, saving you time and material.

Fusion combines all of these (and more, like electronics design) into one seamless program. You don't need separate software for designing, testing, and making. It's all connected.

**The Golden Rule: Think Before You Build** 🧠

The single most important concept in Fusion is **Parametric Modeling**. This might sound complex, but it's like building with intelligent rules.

Think of it this way: Instead of just drawing a box, you give the computer a *recipe* for the box.

- **Rule 1:** The base is a rectangle.
- **Rule 2:** The length of the base is 50mm.
- **Rule 3:** The width of the base is 30mm.
- **Rule 4:** The height is 20mm.

These rules or parameters (**length**, **width**, **height**) define your model. This is the "parametric" part.

Why is this the Golden Rule? Because if you want to change your model, you don't erase and redraw it. **You simply change the rule (the parameter).** If you decide the length should be 70mm instead of 50mm, you just change that one number, and the entire 3D model intelligently updates itself.

This means every design in Fusion starts with a smart 2D **sketch**, which is then used to create a 3D **feature**. We will spend the next few pages mastering this exact process. Always remember: your model is only as good as the rules you give it.

**Page 2: The User Interface (UI) Tour**

When you first open Fusion, the screen can look a little intimidating, like the cockpit of an airplane. But don't worry, it's organized very logically. Think of it like a car's dashboard: everything has its place and is grouped by its function. Let's take a tour of the five main areas you'll be using constantly.

**The Key Areas**

1. **The Toolbar (Top):** This is where you find all your tools—the digital equivalent of your wrenches, saws, and drills. This bar is smart; it changes to show you the most relevant tools for the task you're currently performing. For example, when you're drawing a 2D sketch, it shows you line and circle tools. When you're working with a 3D shape, it shows you tools for pushing, pulling, and cutting. You'll also see a dropdown menu on the far left of this bar to change your **Workspace** (from Design to Manufacture, for instance).

2. **The Browser (Left Side):** If the Toolbar is your tool chest, the Browser is your project's detailed ingredient list. It shows a tree-like structure of everything that makes up your model. This includes your **sketches**, 3D **bodies**, construction planes, and origins. It's incredibly important for organization. You can click the little eye icon ( 👁 ) next to any item in the Browser to hide or show it in the main window.

3. **The Canvas (The Big, Main Area):** This is your stage. The Canvas is the large, open space in the center of the screen where you will build, view, and interact with your 3D models. All the action happens right here.

4. **The ViewCube (Top Right of the Canvas):** This little cube is your camera operator. It controls how you look at your model. You can click on any of its faces (like TOP, FRONT, or RIGHT) to snap to that specific view. You can click and drag the cube to rotate your view freely. There's also a tiny **Home icon** ( 🏠 ) just above it. Clicking this will always return you to the default 3D perspective view, which is incredibly useful when you get lost.

5. **The Navigation Bar (Bottom of the Canvas):** This bar gives you more direct control over your view. The most important tools here are:

   o **Orbit:** Lets you rotate your view around your model.

   o **Pan:** Lets you slide your view up, down, left, or right without rotating.

   o **Zoom:** Lets you move closer to or further away from your model.

   o **Pro Tip:** You can do all of this with a three-button mouse! Hold down the middle mouse button to **Pan**. Hold Shift and the middle mouse button to **Orbit**. Scroll the wheel to **Zoom**. Mastering this will make your workflow ten times faster.

**Page 3: Your First Sketch - The Foundation**

Everything you build in Fusion, from a simple box to a complex engine, starts with a flat, 2D drawing. This drawing is called a **sketch**. Think of it like a blueprint drawn on the ground before you start building the walls of a house. It's the foundation for everything that comes after.

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**Entering the Sketch World**

To begin, you'll find a **Create Sketch** button in the top-left of your Toolbar. It looks like a square with a pencil on it.

When you click it, something important happens: your 3D space disappears for a moment, and Fusion presents you with three orange, see-through rectangles. These are your **Planes**. You can think of them as three pieces of paper floating in space, intersecting at the center: one flat like the floor (the XY-plane), and two standing upright like walls (the XZ and YZ-planes).

Fusion is asking you, "Which surface do you want to draw on?"

At the center of where these planes meet is the **Origin**. It's the (0,0,0) point of your digital universe. It is a very good habit to always start your first sketch connected to this Origin. This anchors your design and prevents it from floating randomly in space.

For now, move your mouse over the plane that looks like the floor (the XY-plane) and click on it.

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**Drawing Your First Lines**

As soon as you select a plane, your view will automatically rotate to look straight down at it, and the Toolbar at the top will change to show you sketching tools. Let's try two of the most basic ones.

- **The Line Tool:**
    - Find the **Line** icon in the toolbar or, even better, press the L key on your keyboard.
    - Click once on your canvas to set the starting point of the line.
    - Move your mouse to a different spot. You'll see a blue line stretching from your starting point.
    - Click again to set the end point. You've drawn a line! You can keep clicking to create a chain of lines. To stop, hit the Esc key.
- **The Rectangle Tool:**
    - Find the **2-Point Rectangle** icon (it's often in a dropdown under Create) or press the R key on your keyboard.
    - Click once to place the first corner of your rectangle.
    - Move your mouse away diagonally. You'll see the rectangle stretch out.
    - Click a second time to place the opposite corner.

Right now, don't worry about making things a perfect size or shape. Just draw a few lines and rectangles to get a feel for it. When you're done experimenting, you can click the **Finish Sketch** button in the Toolbar.

You've now created the fundamental building block of a 3D model.

**Page 4: Sketch Constraints - The Rules of Drawing**

In the last lesson, you drew some lines and rectangles. You might have noticed that you could click and drag them around, changing their angles and positions freely. They were "dumb" shapes. **Constraints** are what make them smart.

Think of constraints as invisible magnets that snap your sketch geometry into place. They are rules that define the *relationships* between different parts of your sketch. If a sketch is a blueprint, constraints are the notes that say, "this wall must be parallel to that wall," or "this corner must always touch that corner."

**Automatic Constraints: Fusion's Helping Hand**

Fusion is smart and tries to predict your intentions. When you are drawing, it will automatically add constraints based on what it thinks you want to do.

For example, if you draw a line that is very close to being perfectly horizontal, Fusion will snap it to be *perfectly* horizontal. You'll see a small grey symbol appear next to the line. This is the **Horizontal constraint** symbol.

Here are a few common automatic constraints to watch for:

- **Horizontal/Vertical:** Locks a line to be perfectly flat or upright.
- **Coincident:** This is a big one. It means "at the same location." It locks a point onto another point, line, or curve. When you snap the start of a new line to the end of an old one, a Coincident constraint is automatically applied.
- **Perpendicular:** Locks two lines at a perfect 90-degree angle to each other.
- **Parallel:** Forces two lines to run in the same direction, like railroad tracks.

**Manual Constraints: Taking Control**

Sometimes, you need to apply a rule after you've already drawn something. This is where the **Constraints Toolbar** comes in. It's usually located on the right side of the main Toolbar when you are in the Sketch environment.

Let's say you drew two lines at random angles and now you want them to be parallel. The process is simple:

1. Find and click the **Parallel** constraint icon in the toolbar.
2. Click on your first line.
3. Click on your second line.

Instantly, one of the lines will rotate to become perfectly parallel to the other. You can do this with many other constraints, like **Equal** (to make two lines the same length) or **Midpoint** (to lock the end of one line to the exact center of another).

A key thing to notice is the color of your sketch lines. **Blue lines are unconstrained** or "undercooked"—they can still be moved around. **Black lines are fully constrained**—their position and geometry are locked in place by rules. Our goal is always to have black lines.

**Page 5: Sketch Dimensions - Giving Size to Your Ideas**

So far, we've used **constraints** to tell our sketch *how* to behave—for example, "this line must be horizontal" or "these two circles must share the same center point." Now, we need to tell our sketch its *size*. This is done with **dimensions**.

If constraints are the grammar that structures your sketch, dimensions are the specific numbers that give it scale. Together, they create a robust and intelligent blueprint.

**The Dimension Tool (Your Digital Measuring Tape)**

Using the dimension tool is one of the most common actions you'll take in Fusion. The best way to access it is by simply pressing the D key on your keyboard. Your cursor will change, indicating you're in dimensioning mode.

Here's how to use it:

1. **Press D** to activate the tool.
2. **To dimension the length of a line:** Click on the line itself. Move your mouse away, and you'll see a dimension tag appear. Click again to place it. A small box will pop up.
3. **Enter a value:** Type in the exact size you want (e.g., 100 for 100 millimeters) and press Enter. The line will instantly resize to that exact length.

You can dimension more than just single lines:

- **Distance between two points or lines:** Click the first item, then click the second item.
- **Angle between two lines:** Click the first line, then the second line.
- **Diameter of a circle:** Click the edge of the circle.

**The Goal: A "Fully Defined" Sketch**

Remember how we said **blue lines** are "undercooked" and **black lines** are "fully cooked"? Dimensions are the final ingredient in that recipe.

A sketch becomes **fully defined** (turning all its lines black) when you have provided enough constraints and dimensions that nothing is left to guesswork. Every point and line is locked in a specific size and position relative to the origin.

Why is this so important?

- **Stability:** A fully defined sketch is predictable. It won't change or break unexpectedly when you make other modifications to your model.
- **Easy Edits:** Remember the Golden Rule of Parametric Modeling? If your sketch is fully defined, you can change any dimension—for example, changing a length from 100mm to 125mm—and the entire sketch will intelligently resize itself, following the rules you set with your constraints.

Your goal for every sketch should be to make it fully black before you click that "Finish Sketch" button.

## Page 6: From 2D to 3D - The Extrude Command

You've done the hard work of creating a smart, robust 2D blueprint. Now for the magic. We're going to take that flat drawing and give it depth, turning it into a solid 3D object. The command we'll use for this is the most fundamental 3D tool in your entire digital workshop: **Extrude**.

Any action that creates or modifies a 3D shape is called a **Feature**. Extrude is your very first, and most important, feature.

## What is Extrusion?

Think of the Extrude command like squeezing toothpaste out of a tube. The shape of the tube's opening (your 2D sketch profile) determines the shape of the stream of toothpaste (your 3D object).

In technical terms, Extrude takes a 2D sketch profile and pushes or pulls it along a straight path to create a 3D body.

## How to Use the Extrude Command

Let's turn that fully defined sketch you made into a solid.

1. **Finish Your Sketch:** If you are still inside the sketch environment (with the grid and sketch tools visible), click the **Finish Sketch** button in the toolbar. This brings you back to the main Solid modeling workspace.
2. **Activate the Tool:** The **Extrude** command is usually the first icon in the Create panel. The easiest way to activate it is to press the E key on your keyboard.
3. **Select a Profile:** An "Extrude" dialog box will appear. Fusion needs to know *what* you want to extrude. It will automatically highlight any **closed profiles** in blue. A closed profile is just a shape made of a continuous loop of lines with no gaps, like a circle or a square. Click inside the shape you want to turn into a solid.
4. **Set the Distance:** Once your profile is selected, you'll see a blue arrow. You have two options:
   - **Drag the Arrow:** Click and drag the arrow to visually pull the shape into 3D.
   - **Enter a Value:** For precision, type an exact distance (e.g., 25mm) into the Distance field in the dialog box.
5. **Confirm the Operation:** The dialog box will show the Operation is set to New Body. This is perfect. It means you are creating your very first solid object. Click **OK**.

Congratulations! You have just completed the fundamental workflow of solid modeling: you created a precise 2D sketch and used it to generate a 3D body. This **Sketch -> Extrude** process is one you will use over and over again.

**Page 7: The Design Timeline - Your Model's History**

At the very bottom of your screen, you'll see a bar with a few icons in it. This unassuming little area is one of the most powerful aspects of Fusion. This is the **Design Timeline**.

Think of the Timeline as your model's interactive history book. Every major action you take—creating a sketch, extruding a shape, cutting a hole—is recorded as an icon, or "feature," in this bar from left to right. It's the step-by-step recipe that created your final design.

For the part we just made, your timeline should be very simple. It will show two icons: one for the **Sketch** and one for the **Extrude** feature that followed.

**The Power of Editing the Past**

The Timeline isn't just for looking at what you've done; it's a time machine that lets you go back and change any decision you made. This is the heart of parametric modeling. You don't have to undo everything and start over if you want to make a change. You just go back to the specific step in the timeline and edit it.

Let's try it.

**To Edit the Original Sketch:**

1. Move your mouse to the Timeline at the bottom of the screen.
2. Find the very first icon, which represents your sketch.
3. **Right-click** on that sketch icon and select **Edit Sketch** from the menu.
4. You are now back inside your original 2D sketch. Double-click on one of the dimensions you created (e.g., the length) and type in a new number. Press Enter.
5. Click **Finish Sketch**.
6. Watch what happens: Your entire 3D model automatically rebuilds itself based on the new dimension.

**To Edit the Extrusion Feature:**

1. Go back to the Timeline.
2. Find the **Extrude** icon (it will be right after the sketch icon).
3. **Right-click** on it and select **Edit Feature**.
4. The same Extrude dialog box you saw before will pop up. Change the Distance value to something new.
5. Click **OK**.

The thickness of your 3D model will instantly update. This ability to non-destructively edit any feature at any point in your design process is what makes Fusion so efficient and powerful. If your model is built correctly, you can change a single dimension in your first sketch and have the entire complex design update perfectly.

## Page 8: More 3D Modeling Tools - Revolve and Sweep

Extruding is perfect for creating shapes that have a straight, linear depth, like blocks, plates, and posts. But what about creating rounded or curved objects like a wheel, a bowl, or a winding pipe? For that, we need different tools. Let's explore two more essential 3D creation features: **Revolve** and **Sweep**.

### The Revolve Command: The Digital Potter's Wheel 🏺

The **Revolve** command creates a 3D shape by spinning a 2D sketch profile around a central line, known as an axis.

**The Analogy:** Imagine a potter's wheel. A potter places a lump of clay and shapes one side of it. When the wheel spins, that one profile shape forms a complete, symmetrical pot. The Revolve tool works the exact same way.

**How to Use It:**

1. **Create a sketch.** This sketch needs two things: a **closed profile** (the cross-section of your object, like half of a donut shape) and a separate **line** to serve as the **Axis** of revolution.
2. Exit the sketch and find the **Revolve** command in the Create panel.
3. The dialog box will ask for two selections:
   - **Profile:** Click the closed shape you want to spin.
   - **Axis:** Click the line you want to spin it around.
4. As soon as you select both, a 3D preview will appear. By default, it revolves a full 360 degrees, but you can change this.
5. Click **OK**.

You've now created a revolved part. This technique is perfect for anything that is symmetrical around a center point, like wheels, rings, vases, and doorknobs.

### The Sweep Command: Shapes Along a Path 🪢

The **Sweep** command takes a 2D profile and pushes it along a defined curve or line, called a path.

**The Analogy:** Think of the Extrude command, but instead of going in a straight line, it follows a winding road. You define the shape of the car (the profile) and the road it has to drive on (the path).

**How to Use It:**

1. This feature requires **two different sketches**.
   - **Sketch 1 contains the Path:** This is the line or curve that your shape will follow (e.g., an S-curve).
   - **Sketch 2 contains the Profile:** This is the closed shape you want to drag along the path (e.g., a small circle). This sketch must be created on a plane that is perpendicular to the start of your path.
2. Exit the sketch and find the **Sweep** command in the Create panel.
3. The dialog box will ask for the two key ingredients:
   - **Profile:** Select your closed shape (the circle).
   - **Path:** Select the curve you drew (the S-curve).
4. A preview will show your circle swept along the S-curve, creating a pipe-like shape.
5. Click **OK**.

Sweep is the go-to tool for creating things like pipes, tubes, springs, and handrails.

Creating the main shape of your model is only the first step. The real art and professionalism come from refining that shape. This is done with **modification features**. These are tools that don't create new bodies from scratch, but rather alter and add detail to the bodies you already have. Let's look at three of the most common ones: **Fillet**, **Chamfer**, and **Shell**.

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### The Fillet Command (Rounding Edges)

A **Fillet** (keyboard shortcut: F) is a command that creates a rounded corner on an edge of your model.

- **What it does:** It takes a sharp, 90-degree edge and makes it smooth and curved.
- **Analogy:** Think of taking a piece of sandpaper to a sharp wooden corner to make it smooth and safe to touch.
- **Why use it?** Fillets are used for aesthetics (they look nice), for safety (sharp edges are dangerous), and for strength. Sharp internal corners can be weak points; rounding them with a fillet distributes stress and makes the part more durable.
- **How to use it:** Press the F key, click on one or more sharp edges of your model, and then enter a **radius** (e.g., 2mm) to define how round you want the edge to be.

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### The Chamfer Command (Beveling Edges)

A **Chamfer** is a command that creates a flat, angled surface on an edge of your model, also known as a bevel.

- **What it does:** It takes a sharp edge and cuts it away at an angle (typically 45 degrees).
- **Analogy:** If a fillet is sanding, a chamfer is like taking a plane and slicing the corner off cleanly.
- **Why use it?** Chamfers are used for a clean, technical look and also for function. A chamfer on the edge of a hole can help guide a peg or bolt into it easily.
- **How to use it:** Select **Chamfer** from the Modify panel in the toolbar, click on the edge you want to change, and enter a **distance** (e.g., 2mm) to define the size of the flat.

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### The Shell Command (Hollowing Out Parts)

The **Shell** command is used to hollow out a solid body, leaving a thin wall of a specific thickness.

- **What it does:** It scoops out the inside of your model, turning a solid block into an open container.
- **Analogy:** It works exactly like carving a pumpkin. You cut open one face and then remove all the material from the inside, leaving the outer wall.
- **Why use it?** This is essential for creating things like enclosures, cases, or containers. It saves a tremendous amount of material and weight.
- **How to use it:** Select **Shell** from the Modify panel. First, select the **face** you want to remove (the opening of your container). Then, specify the desired **wall thickness**. Fusion does the rest, hollowing out the entire part.

Each of these modifications will add a new icon to your Design Timeline, allowing you to go back and edit the radius, distance, or thickness at any time.

**Page 10: Creating Holes and Patterns**

As you get more comfortable with modeling, you'll find yourself needing to create more complex and repetitive features. Fusion has specialized tools that make these tasks both easier and more intelligent. Let's look at the dedicated **Hole** command and the incredibly efficient **Pattern** tools.

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**The Hole Command: An Intelligent Cut**

You might be thinking, "Can't I just sketch a circle and Extrude-cut it to make a hole?" Yes, you can. But using the dedicated **Hole** command (keyboard shortcut: H) is a much smarter way to work.

A simple extruded cut is just a geometric void. A feature created with the Hole command contains rich manufacturing information. You can define specific types of holes for real-world hardware:

- **Simple:** A straight-through hole.
- **Counterbore:** A hole with a larger-diameter, flat-bottomed recess at the top, designed to fit the head of a socket screw so it sits flush with the surface.
- **Countersink:** A hole with a cone-shaped recess at the top, designed for a flat-head screw to sit flush.

**How to Use It:**

1. The best practice is to first create a **sketch** on the face where you want the hole, and use the **Sketch Point** tool to precisely place the center of your future hole. Then, finish the sketch.
2. Press the H key to activate the Hole command.
3. In the dialog box, select the **Sketch Point** you just made for placement.
4. Now you can define everything about the hole: its **diameter**, its **depth**, and its **type** (Simple, Counterbore, or Countersink). You can even add threads.
5. Click **OK**. You now have an intelligent hole feature that can be easily identified on a technical drawing.

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**The Pattern Tools: Work Smarter, Not Harder**

Imagine you need to create a grid of 16 holes. You could create each one individually, but that would be slow and create a very cluttered Design Timeline. The smart way is to create *one* perfect hole, and then use a **Pattern** tool to copy it.

**The Rectangular Pattern:** This tool copies a selected feature (or body) in a grid of rows and columns.

**How to Use It:**

1. Go to the Create menu, then Pattern, and select **Rectangular Pattern**.
2. A dialog box will appear. It asks for three main things:
   - **Objects:** What do you want to copy? You can select the **Hole feature** directly from your Timeline at the bottom of the screen.
   - **Directions:** You need to tell Fusion which way to make the grid. Click the "Directions" selector and then click on a straight edge of your model to define the X-axis, and another edge for the Y-axis.
   - **Quantity and Distance:** Enter how many copies you want and the spacing between them for each direction.
3. Click **OK**. Fusion will instantly create all the copies.

The best part? Because this is parametric, if you go back and edit the *original* hole feature (for example, changing its diameter), all the patterned copies will update automatically. There is also a **Circular Pattern** tool for creating copies arranged in a circle, like the lug nuts on a wheel.

**Page 11: Introduction to Assemblies**

Everything we have designed so far has been a single, self-contained part. However, most products in the real world—from a simple ballpoint pen to a complex car engine—are **assemblies** of multiple parts that fit and move together.

Fusion is exceptionally good at handling assemblies. But to work with them, we must first understand the single most important concept in assembly modeling: the difference between a **Body** and a **Component**.

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**Bodies vs. Components: A Critical Difference**

- **What is a Body?** A **Body** is any single, continuous 3D shape. The block you extruded and the wheel you revolved were both Bodies. Think of Bodies as individual pieces of wood on a workbench. You can have several pieces, but they are just lying there; they don't have any defined relationship and can't be told how to move relative to each other. In the Browser on the left, they are all grouped together in a single "Bodies" folder.

- **What is a Component?** A **Component** is like a special container that holds one or more bodies, plus all the sketches and construction geometry needed to make them. A Component represents a real-world, individual part. It has its own identity, its own origin, and its own coordinate system.

**The Golden Rule of Assemblies:** You cannot create motion or define relationships (called Joints) between Bodies. You can *only* create joints between **Components**.

Therefore, if you intend to design something that has multiple parts that need to connect or move, you *must* design each part as a separate Component.

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**How to Work with Components**

Working with components is straightforward if you follow one simple rule.

**Rule #1: Activate First.** Before you start building a specific part, you must tell Fusion which component you are working on. You do this by **activating** it. In the Browser, simply click the small radio button (a dot) that appears next to the component's name. When a component is active, everything else in your design will become slightly transparent, helping you focus on the part at hand.

The best practice is to create a new, empty component for each part *before* you start designing it.

**How to Create a New Component:**

1. Look at the Browser on the left side of your screen.
2. **Right-click** on the very top-level item in the tree (the name of your entire design file).
3. From the dropdown menu, select **New Component**.
4. Give it a meaningful name (e.g., "Axle" or "Casing") and click OK.

You now have a proper, self-contained component, ready for you to create sketches and features inside it.

**Page 12: Assembling with Joints**

Now that you understand how to create separate **Components**, the next step is to define how they connect and interact. Are they glued together? Does one part spin inside another? Does a piece slide back and forth? The tool we use to define all these relationships is called a **Joint**.

**What are Joints?**

In the real world, a component floating in space has six ways it can move, known as "degrees of freedom": it can slide along the X, Y, and Z axes, and it can rotate around the X, Y, and Z axes. A **Joint** connects two components and removes some or all of these degrees of freedom.

**Analogy:** Think of joints as digital versions of real-world connections. A **Rigid** joint is like a weld or glue, locking two parts together completely. A **Revolute** joint is like a door hinge, allowing only rotation. A **Slider** joint is like a drawer slide, allowing only linear motion.

Instead of just restricting motion, Fusion's joints define the *allowed motion* between components.

**Applying a Rigid Joint**

The most common and fundamental joint is the **Rigid** joint. It locks two components together, removing all six degrees of freedom. Let's assume you have designed two components right next to each other, exactly where they should be. This is called "in-place" or "top-down" design. The easiest way to connect them is with an **As-Built Joint**.

**How to use the As-Built Joint:**

1. Make sure your main assembly is activated in the Browser (click the radio button next to the top-level name).
2. Go to the **Assemble** panel in the toolbar and select **As-Built Joint**.
3. The dialog box will ask you to select the two components you want to join. You can click on them in the main Canvas or select them from the Browser list.
4. After selecting both, a new menu for Motion type will appear.
5. Select **Rigid**. You'll see the components do a quick "shiver" animation to confirm they are locked. A joint icon will appear.
6. Click **OK**.

Now, if you try to click and drag one of those components, the other will move with it perfectly. They are now treated as a single, rigid group.

While Rigid is the most common, other motion types like **Revolute** (for hinges), **Slider** (for sliding parts), and **Ball** (for ball-and-socket joints) allow you to create complex mechanisms with realistic motion. Each one you create will appear in the "Joints" folder in your Browser.

**Page 13: The Sculpt Environment (T-Splines)**

So far, every model we've built has been based on precise sketches, dimensions, and rules. This is called **Solid Modeling**, and it's perfect for mechanical, engineered parts. But what if you want to design something with smooth, flowing, organic curves, like a computer mouse, a car body, or a bicycle helmet?

For this, Fusion has a completely different and more artistic toolset called the **Sculpt Environment**, which uses a technology called **T-Splines**.

**Analogy:** If Solid Modeling is like building with precise LEGO bricks, the Sculpt Environment is like shaping a lump of digital clay.

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**Entering the World of Digital Clay**

To begin sculpting, you don't change your workspace. Instead, within the Design workspace and under the Solid tab, you'll find an icon of a purple cube called **Create Form**. Clicking this button will transport you into the Sculpt environment. The toolbar will turn purple, and you'll see a whole new set of tools dedicated to pushing and pulling shapes.

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**How Sculpting Works**

The process usually starts by creating a primitive shape, like a **Box**, **Sphere**, or **Cylinder**, from the purple Create panel. This gives you your initial "lump of clay."

This shape will be covered in a control cage made of **faces**, **edges**, and **vertices** (points). The primary tool you will use is **Edit Form**.

When you activate Edit Form, you can click on any face, edge, or vertex. A manipulator gizmo with arrows and squares will appear. You can then literally push, pull, and rotate that part of the cage, and the entire surface of the model will smoothly and fluidly deform around your change. You can grab a single point and pull it to create a spike, or select a whole face and scale it up to create a bulge. It's an incredibly intuitive and creative process.

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**From Form back to Solid**

The power of Fusion is in combining these two modeling worlds. Once you have sculpted your organic shape and are happy with it, you click the **Finish Form** button in the toolbar.

This action converts your fluid, purple T-Spline body into a standard, solid 3D body—the same kind you get from an Extrude command. Once it's a solid, you can use all your familiar tools on it. You can cut precise holes into your sculpted shape, use the Shell command to hollow it out, or add a filleted edge. This allows you to create highly ergonomic and aesthetically pleasing products that also meet precise mechanical requirements.

**Page 14: Introduction to Manufacturing (CAM)**

So far, our entire journey has been in the digital realm—designing, shaping, and assembling parts on the screen. This process is **CAD**, or Computer-Aided Design. Now, let's briefly touch on how we can prepare our digital design to be made into a physical object. This is **CAM**, or Computer-Aided Manufacturing.

**What is CAM?**

CAM is the process of using software to create instructions that computer-controlled machines can understand.

**Analogy:** Imagine you've designed a beautiful part on your computer. CAM is like writing a very precise, step-by-step instruction manual for a robot arm with a drill bit. This manual tells the robot exactly where to move, how fast to go, and how deep to cut to carve your part out of a solid block of aluminum.

These instructions are called **toolpaths**, and the final instruction file, written in a language called **G-code**, is what you load into a CNC (Computer Numerical Control) machine, like a mill or a router.

**The Manufacture Workspace**

In Fusion, you can switch from the Design workspace to the **Manufacture** workspace using the dropdown menu at the top-left. The environment will change, giving you tools to prepare your part for machining. The process boils down to three main steps:

1. **Setup:** The first step is to tell Fusion about the real world. You define your **stock**, which is the raw block of material you are starting with. You also tell Fusion how your finished part sits inside that block and where the machine's zero point, or origin, is.

2. **Create Toolpaths:** This is the core of CAM. A **toolpath** is the exact path the cutting tool will take to remove material. You choose different strategies for different tasks. For example, you might use a "2D Adaptive Clearing" toolpath to quickly remove large amounts of material, and then a "Contour" toolpath to create a clean, finished wall. For each toolpath, you select a specific cutting tool from a library (like a 6mm Flat End Mill) and define its speeds and feeds.

3. **Simulate and Post-Process:** Before you ever risk crashing a real, expensive machine, you **simulate**. Fusion provides an animation that shows your chosen tool cutting your virtual stock material. This is critical for catching errors. Once you're confident the simulation is perfect, you **post-process** it. This is the final step where Fusion translates your toolpaths into the specific G-code file that your machine can read.

The incredible power of Fusion is that if you go back to the Design workspace and change the size of your part, the Manufacture workspace will flag the toolpaths as out-of-date, allowing you to quickly update them to match the new design.

**Page 15: Creating 2D Drawings**

We have come full circle. Our journey started with a 2D sketch, which we used to create a 3D model. Now, we will take that 3D model and use it to generate a formal 2D technical drawing. While a 3D model is fantastic for visualization, a clear, precise 2D drawing is often the final document used for manufacturing, inspection, and patent applications.

Fusion has a dedicated **Drawing Workspace** for creating these professional documents directly from your designs.

**Entering the Drawing Workspace**

To start, you navigate to the File menu in the top-left corner, select New Drawing, and then **From Design**. A dialog box will appear, allowing you to choose your drawing standard (like ASME for the US or ISO for international) and your sheet size. Once you click OK, you'll be taken to a new environment that looks like a piece of paper.

**Placing Views**

Your first task is to place a **Base View**. This is the main, primary view of your part, typically the Front view. You can set its scale and position on the sheet.

Once your base view is placed, the magic happens with the **Projected View** tool.

1.  Click the Projected View command.
2.  Click on your base view to select it.
3.  Move your mouse directly above the base view and click again. A Top view is instantly created and perfectly aligned.
4.  Move your mouse to the right of the base view and click. A Right side view is created, also perfectly aligned.
5.  Move your mouse to the top-right corner diagonally, and you can place a shaded, 3D Isometric view.

You can also create more advanced views, like a **Section View** to show an imaginary slice through the middle of your part, or a **Detail View** to show a magnified bubble of a small, intricate area.

**Dimensions and Annotations**

A drawing is useless without dimensions. Using the **Dimension** tool, you can add all the critical measurements that a manufacturer would need. You click on edges and points to place lengths, diameters, radii, and angles. The goal is to provide a complete set of instructions so there is no ambiguity about the part's geometry.

The title block in the corner of the sheet will automatically be filled with information from your model, like the project name, author, and date.

**The Power of Association**

The most powerful feature of the Drawing workspace is its direct link to the 3D model. The drawing is **associative**. If you go back to the Design workspace and change a dimension—for example, making your part 10mm longer—the Drawing will automatically update. The views will stretch to the new size, and all the dimension values will change to match. This integration saves countless hours and prevents critical errors.

**Conclusion**

Congratulations! You have just journeyed through the entire core workflow of Autodesk Fusion. You started with a simple 2D sketch (Page 3), gave it rules with constraints and dimensions (Pages 4-5), and brought it to life in 3D

with features like Extrude and Revolve (Pages 6-8). You learned how to refine your model (Page 9), add intelligent features (Page 10), and assemble it with other parts (Pages 11-12). You even explored the world of organic sculpting (Page 13) and got a glimpse of how to prepare your part for manufacturing (Page 14) and document it for others (Page 15).

This foundation is your key to unlocking the incredible potential of digital design and manufacturing. Keep practicing these fundamentals, and you will be well on your way to creating anything you can imagine.