

This notebook is here as a resource to help you learn Onshape
how can i create a cactus shape in onshape?

Creating a 3D model of a complex shape like a cactus in Onshape relies on the core principles of **Sketch-Based Modeling**, combining 2D sketches with various 3D features to build the final part. Depending on the specific type of cactus (e.g., simple cylinder, segmented, or highly organic), you would use different foundational features or advanced surfacing techniques.

Here is an overview of the steps and relevant Onshape tools for modeling a cactus shape:

1. Starting the Part Design

1. **Create a New Document:** Begin by creating a new document in Onshape. By default, this opens an empty **Part Studio** and an **Assembly** tab.

2. **Start a Sketch:** Select the **Sketch** tool (or use the shortcut **Shift + S**). You must select a plane (Front, Top, or Right) or a planar face of an existing part to define the 2D sketching surface.

3. **Orient the View:** Press the **N key** to automatically orient the camera to view the sketch plane directly (normal to the screen). You can also press the **P key** to hide the default planes for a cleaner workspace.

2. Modeling the Main Cactus Body

The primary body of the cactus (the main stem or base) can be created using one of the four foundational features:

Feature	Application for Cactus Shapes	Mechanics
Revolve	Ideal for cacti with radial symmetry , such as columnar cacti (like a saguaro or barrel cactus).	Sketch a 2D half-profile of the cactus on a plane. Then, revolve this profile around a central line (axis of revolution) to create the 3D shape.
Loft	Useful for shapes that taper or blend between different cross-sections, like a plant that transitions from a rectangular base to a rounded top.	Define two or more unique 2D sketch profiles on planes offset from each other. Loft smoothly connects these profiles to create a blend.
Extrude	Best for simple shapes or bases with a constant cross-section .	Sketch the outline of the cactus base (e.g., a circular or polygonal shape). Use Extrude (Shift + E) to give it height or thickness.

Sweep	Useful if an arm or segment curves along a path while maintaining a constant profile.	Create two sketches, usually perpendicular to each other: one for the cross-section (the profile) and one for the path (the curve).
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Remember to ensure your 2D sketch used for the 3D feature is a **closed contour** (indicated by shading) to create a solid body.

3. Adding Details and Features (Ribs, Arms, and Spines)

Once the main body is created, you can add or remove material to refine the shape, such as adding ribs or side arms:

- **Creating Indentations/Ribs:** To simulate the characteristic vertical ribs or pleats found on many cacti, you can remove material. Create a new sketch on the face of your part. Sketch the profile of the groove or indentation. Use the **Extrude** feature, changing the operation type from "Add" to **"Remove"** to cut the material away.
- **Adding Arms/Protrusions:** To add side arms, sketch the cross-section of the arm on a flat face, then use **Extrude** with the **"Add"** option to merge it with the main body.
- **Creating Repeating Features:** If your cactus has multiple evenly spaced ribs or arms, model one instance and then use a **Feature Pattern** (such as **Circular Pattern**) to duplicate it around a central axis.
- **Smoothing Edges:** Use the **Fillet** tool (**Shift + F**) to round sharp corners and smooth transitions between features, making the cactus appear more natural.

4. Advanced Modeling for Organic Shapes

If your cactus design requires highly complex or freeform surfaces that are difficult to achieve with simple geometric features, you may use **Surface Modeling** techniques:

1. **Wireframe Foundation:** Surface modeling often begins by constructing a precise **wireframe layout** using 2D sketches and 3D curves. You might use custom tools like **Control Point Curve** for intricate shapes.

2. **Building Surfaces:** Use features like **Boundary surface** or **Loft** to build the surface model face by face based on your curves.

3. **Trimming and Blending:** Build surfaces larger than necessary (to theoretical edges) and then trim them back. Use **Bridging Curve** for building smooth transitions and blends between complex surface patches.

Learning Resources

If you encounter difficulties or wish to learn more about specific tools used for complex modeling, Onshape offers many resources:

- **Learning Center:** Provides self-paced courses and documentation, including an **Introduction to Part Design** and **Part Design Using Part Studios**.

- **Tech Tips & Videos:** Access technical tips and videos from Onshape experts.

- **Forums:** The Onshape Forums are available to ask questions and discuss modeling challenges with the community.

What is a mechanical drawing, and what features should it include?

Based on the sources related to Onshape, a mechanical drawing (often referred to simply as a "Drawing") is a specific data container and output type used for detailing parts and assemblies.

What is a Mechanical Drawing?

A mechanical drawing, within the context of the Onshape platform, is a type of tab or document output used for technical purposes.

- **Format and Location:** A **Drawing** is one of the three primary types of tabs (alongside **Part Studio** and **Assembly**) that can be generated within an Onshape document.
- **Source Material:** You can create mechanical drawings from within **Onshape Part Studios** and **Assemblies**, or of entire Part Studios. When creating a drawing of an Assembly, it automatically includes the solid bodies and parts.
- **File Type:** All Onshape drawings are based on the **.DWG** (drawing database) file format, and the **.DXF** (Drawing Interchange File) format is also supported.

Features a Mechanical Drawing Should Include

The primary purpose of a drawing tab is to provide a 2D representation of the design that can be **annotated**. Drawings rely on dimensions, notes, and specific views to convey design intent.

1. Views and Geometry:

- A drawing can display various views, including a **Projected view** (accessible via the shortcut **P**).
- Geometric entities like a **Line** (**L** shortcut) can be included.
- Drawings can utilize different templates (such as inch or millimeter templates) for displaying dimensions.

2. Annotations and Dimensions: The Drawings toolbar includes tools for detailed annotations and measurements:

- **Linear dimension** (**D**).
- **Radial dimension** (**Shift+R**).
- **Diameter dimension** (**Shift+D**).
- **Note annotation** (**N**).
- The ability to snap to the **midpoint and quad points** on arcs and circles for selection during dimension commands (**Shift+Q**).
- The **Maximum or minimum dimension tool** (**Ctrl+M** in the feature set list, or **m** in the shortcut list).

3. Data and Management Tools:

- A key feature often included in drawings is the **Bill of Materials (BOM) table**. These BOM tables can be inserted into your drawings.
- Drawings must be kept up-to-date. You can **Update all views and properties** on each sheet of the drawing using the shortcut **Ctrl+Q** to reflect any modifications made in the associated part or assembly file.
- Users can manage multiple sheets, accessing the **next sheet** (**PgDn**), **previous sheet** (**PgUp**), **first sheet** (**Home**), or **last sheet** (**End**).
- The drawing information can be viewed by accessing the **Properties** dialog, where meta data for the entire Drawing can be provided.

what should a bill of materials include for a 3d-printed project? I currently have only plastic filament, would that be detailed enough?

A **Bill of Materials (BOM)** is a core element of data management in platforms like Onshape, designed to define the required components of a design. For a 3D-printed project, the BOM should detail all necessary components and materials, and simply listing "plastic filament" may not be sufficiently detailed based on the capabilities available in the CAD system.

Features a Bill of Materials Should Include

The BOM is typically generated automatically from an Onshape **Assembly**. Whether your project is a single part or an assembly of several parts, the BOM should contain comprehensive details to fully define the components.

Default Properties: Onshape BOMs include a default set of columns/properties. The essential properties usually displayed for parts and subassemblies are:

- Item number
- Quantity
- Part number
- Description

Material and Non-Geometric Details: For a fully detailed project, the BOM should account for materials, especially as these are assigned within the CAD software to define physical properties:

1. **Material Assignment:** When modeling a part, a material is typically assigned to the solid body (e.g., using a right-click menu option like "assign material"). Assigning a material is necessary to calculate **mass properties**. If your part is designed using ABS, for example, that material should be explicitly referenced.
2. **Non-Geometric Items:** The BOM can include non-geometric items (items not modeled in CAD) that are required for the final product. Examples of such items include **glue, tape, paint, or thread locker**. If your 3D print requires post-processing items (e.g., sanding material, specific adhesive for assembly), these should be explicitly listed.
3. **Vendor Information:** You can add specific vendor information to the part's properties through the BOM table, which helps track where materials or components are sourced.
4. **Custom Properties:** You have the ability to define **Custom Properties** and incorporate them as columns into the BOM. For 3D printing, custom properties might be necessary to detail specifics beyond the material type itself.

Is "Plastic Filament" Detailed Enough?

Listing only "plastic filament" is generally **not detailed enough** for a comprehensive BOM, especially when dealing with production or assembly:

- **Specificity of Material:** The CAD system requires assignment of a specific material (e.g., 1060 aluminum alloy or ABS) to calculate mass accurately. Simply calling it "plastic" is too vague, as filament comes in various types (PLA, ABS, PETG, etc.), each having different mechanical and mass properties.

- **Detailed Description/Custom Properties:** Even if the part material is listed, the BOM should ideally include specifics relevant to printing, such as:

- **Filament Diameter:** (e.g., 1.75mm)
- **Color:** (If necessary for identification or aesthetics)
- **Estimated Weight/Length Used:** (Calculated from the mass properties of the part, converted to grams/kilograms/pounds of the required material).

- If your project is an **Assembly** (meaning it contains multiple distinct components, even if all are 3D printed), the BOM must list **each unique part** individually, along with its specific quantity. In summary, while "plastic filament" covers the basic input material, a complete BOM should specify the exact **material type** (which is assigned to the part in the CAD environment to determine properties) and potentially leverage **Custom Properties** to include print-specific details like color or post-processing supplies.

how can i mark something in a mechanical drawing as increasing incrementally?

Based on the provided sources regarding Onshape's capabilities in creating mechanical drawings, there is **no explicit tool or feature mentioned that specifically allows you to mark a dimension as "increasing incrementally"** (such as indicating a series of steps or a variable dimension using notation like "TYP" or stepped values).

However, you can utilize the standard annotation tools available in the Drawing environment to convey such information manually:

1. **Note Annotation (N):** You can use the **Note annotation tool** (shortcut **N**) to manually add text and specify that a dimension is incremental, typical, or variable.

2. **Dimensioning Tools:** You can dimension one instance using the **Linear dimension tool** (shortcut **D**) and then use the Note tool to indicate the necessary increments or sequence for manufacturing.

3. **Maximum/Minimum Dimension:** While not for incrementing, the **Maximum/minimum dimension tool** (shortcut **Ctrl+M** or **m**) is available for specific dimensional documentation needs within a drawing.

In summary, the sources detail the general tools for creating 2D drawings and annotations, but if a drawing requires a specialized convention for showing incremental dimensions (like those used in patterned geometry), it would typically need to be communicated through a **Note annotation** or by detailing each dimension individually.