Introduction to Onshape and Sketching

Curriculum Guide

Week 1

07/03 - 07/09

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Announcements

This week, we will be going through:

- Onshape Setup
    - Account creation
    - Navigating the menu
    - Creating the first part
    - Navigating the toolbar basics
    - How to orient the screen (tips & tricks)
- Sketching in Onshape

Reminders:

- Watch the Week 1 video on sketching in Onshape here
- Refer to [this link](this%20link) for instructions on downloading and setting up Onshape
- Assignment 1 due March 12, 11:59 pm
- Mentor office hours have been emailed to you

Week 1 Learning Goals

- Basic functions of sketching
    - Planes/reference geometry
    - Types of sketch tools
    - Types of sketch lines
    - Dimensioning
    - Patterns
    - Mirroring
- Navigate through the Onshape user interface
- Understand how a combination of sketch functions can be used to create complex shapes

Essential Onshape Shortcuts and Commands

These are some of the basic commands/keys to use when sketching. For a complete list of Onshape shortcuts, please see here:

https://cad.onshape.com/help/Content/shortcut_keys.htm. Note: use of a mouse is *highly recommended* when using Onshape and any other CAD software!

Navigation:

Left Mouse Click: Select

Left Mouse Drag: Select within box

Right Mouse Drag: Rotate view

Middle Mouse Drag: Pan view

Scroll: Zoom in/out

n: View normal to sketch

f: Zoom to fit

Design:

Esc: Exit menus, tools, etc.

CTRL + c: Copy

CTRL + v: Paste

CTRL + x: Copy and cut

CTRL + z: Undo

Basic Functions of Sketching in Onshape

Sketching is the fundamental process of creating 2D geometry that serves as the foundation for 3D models in Onshape. Here are some basic functions of sketching in Onshape:

a. *Creating a new sketch:* To create a new sketch, select a plane or face in the Feature List where you want to sketch. Right-click on it and choose "Create Sketch."

b. *Selecting sketch planes*: Onshape provides various options for selecting sketch planes, such as faces, planes, or reference geometry. You can also create your own reference planes.

c. *Sketching tools overview:* Onshape offers a range of sketching tools, including lines, rectangles, circles, arcs, ellipses, splines, and more. These tools allow you to create different geometric shapes.

d. *Modifying sketch entities:* You can modify sketch entities by moving, rotating, scaling, trimming, extending, or filleting them. These modification tools help in refining your sketch geometry.

e. *Exiting the sketch:* Once you have finished sketching, click the green checkmark (✅) to exit the sketch mode. Onshape will create a sketch feature in the Feature List based on your sketch.

Planes and Reference Geometry in Onshape

Planes and reference geometry play a crucial role in creating sketches and models in Onshape. They provide the foundation for sketching and allow for accurate positioning and alignment of sketch entities.

Creating Planes:

In Onshape, you can create planes in different orientations to serve as sketching surfaces or reference planes for other operations. Here's how to create planes:

a. Selecting Existing Planes:

- When creating a sketch, you can select existing faces or planes as the sketching plane.
- Simply click on the desired face or plane to set it as the sketching plane.

b. Creating Reference Planes:

- Onshape allows you to create additional reference planes to aid in sketching or modelling.
- To create a reference plane, go to the toolbar and click on the "Reference" tool.
- Choose the desired plane type, such as offset, parallel, perpendicular, or angled plane.
- Specify the required parameters, such as distance or angle, to define the position and orientation of the plane.

Using Reference Geometry:

Reference geometry in Onshape provides additional points, lines, or planes that can be used for constraining and positioning sketch entities accurately. Here are some common types of reference geometry:

a. Reference Points:

- Reference points are used as fixed locations or constraints in sketches.
- They can be created at specific coordinates or by snapping to existing geometry.

b. Reference Axes:
- Reference axes are used to define rotational directions or symmetry in sketches.
- They can be created by specifying endpoints or by snapping to existing geometry.

c. Reference Coordinate Systems:
- Reference coordinate systems define the origin and orientation of a specific coordinate system within a sketch or model.
- They are useful for positioning and orienting sketch entities precisely.

d. Reference Planes:
- Reference planes can be created perpendicular, parallel, or at an angle to existing planes.
- They provide additional surfaces for sketching or serve as references for other operations.

Constraining Sketch Entities to Reference Geometry:

When sketching in Onshape, you can use reference geometry to constrain and align sketch entities. Here are some examples:

a. Coincident Constraint:
- The coincident constraint aligns a sketch entity to a reference point or another sketch entity.
- It ensures that the selected entities share the same position.

b. Horizontal/Vertical Constraint:
- The horizontal or vertical constraint forces a sketch entity to align horizontally or vertically with a reference line or axis.

c. Parallel/Perpendicular Constraint:
- The parallel or perpendicular constraint aligns sketch entities or lines parallel or perpendicular to each other or to a reference line or axis.

d. Tangent Constraint:

- The tangent constraint ensures that a sketch entity, such as a line or an arc, touches another entity tangentially.

e. Symmetry Constraint:

- The symmetry constraint maintains symmetry between sketch entities by creating mirrored copies or enforcing equal distances.

Types of Sketch Tools in Onshape

Onshape offers a range of sketch tools to create different geometric shapes. Here are some commonly used sketch tools:

a. ***Line tool:*** The line tool allows you to draw straight lines between two points or create construction lines.

b. ***Rectangle tool:*** You can use the rectangle tool to draw rectangles or squares. It offers options for creating rectangles by specifying dimensions or corner points.

c. ***Circle tool:*** The circle tool enables you to draw circles by specifying the center point and radius or diameter.

d. ***Arc tool:*** With the arc tool, you can create circular arcs by specifying the start point, endpoint, and center point.

e. ***Ellipse tool:*** The ellipse tool allows you to draw ellipses by specifying the center point and major and minor axes.

f. ***Splines:*** Splines are smooth curves that can be created using control vertices or by sketching approximate curves through multiple points.

Types of Sketch Lines in Onshape

In addition to the basic sketch tools, Onshape provides different types of sketch lines to enhance your sketches. Here are a few examples:

a. **_Construction lines:_** Construction lines are sketch entities used as references or guides and do not contribute to the final geometry. They can be used for positioning other sketch entities.

b. **_Centerlines:_** Centerlines help define the centers of circular or symmetrical geometry. They can be added to circles, arcs, or lines.

c. **_Tangent lines:_** Tangent lines are lines that touch curves or circles at a single point without crossing them.

d. **_Perpendicular lines:_** Perpendicular lines are lines that intersect at a 90-degree angle.

e. **_Parallel lines:_** Parallel lines are lines that maintain an equal distance from each other throughout their length.

Dimensioning in Onshape

In Onshape, dimensioning allows you to specify the size and position of sketch entities and features accurately. When adding dimensions, you can also specify the units of measurement.

Adding Dimensions:

1. To add dimensions to sketch entities or features:
2. Select the sketch entities or features you want to dimension.
3. Go to the toolbar and click on the "Dimension" tool.
4. Click on the sketch entities or features to apply dimensions.
5. Onshape will automatically display the dimension value.

Changing Units:

Onshape allows you to change the units of measurement for dimensions. Follow these steps:

1. Open the document that contains the dimension you want to modify.
2. Select the dimension you wish to change.
3. In the Property Manager on the right side of the screen, locate the "Units" section.
4. Click on the drop-down menu to choose the desired units. Onshape offers a wide range of unit options, including metric and imperial units.
5. After selecting the units, the dimension value will automatically update to reflect the new unit.

Custom Units:

If you need to use custom units that are not available in the standard unit options, you can create custom unit systems in Onshape. Here's how:

1. Open the document that contains the dimension you want to modify.
2. Select the dimension you wish to change.
3. In the Property Manager, click on the drop-down menu in the "Units" section.
4. Scroll to the bottom of the unit options and choose "Custom Unit System."

5. In the Custom Unit System dialog box, you can define your own units by specifying the conversion factors and names for each unit.

6. Click "OK" to save the custom unit system and apply it to the selected dimension.

Changing Units for the Entire Document:

If you want to change the units for the entire document, including all dimensions and measurements, you can do so in the Document Settings. Here's how:

1. Open the document for which you want to change the units.

2. In the top toolbar, click on the document name and select "Document Settings" from the drop-down menu.

3. In the Document Settings dialog box, navigate to the "Units" tab.

4. Choose the desired units from the available options or create a custom unit system.

5. Click "OK" to save the changes and apply the new units to the entire document.

Patterns in Onshape

Patterns in Onshape allow you to replicate sketch entities or features multiple times, creating arrays and repeating patterns. They are useful for quickly generating complex geometries and designs.

Linear Pattern:

The linear pattern creates a linear array of sketch entities or features along a specified direction and distance. Here's how to use the linear pattern:

    a.   Select the sketch entities or features you want to pattern.

    b.   Go to the toolbar and click on the "Linear Pattern" tool.

    c.   Choose the direction and specify the distance between instances.

    d.   Set the number of instances or use the spacing option to define the total length of the pattern.

    e.   Adjust any additional settings, such as orientation or flipping options.

    f.   Click "OK" to create the linear pattern.

Circular Pattern:

The circular pattern creates a circular array of sketch entities or features around a center point. Here's how to use the circular pattern:

    a.   Select the sketch entities or features you want to pattern.

    b.   Go to the toolbar and click on the "Circular Pattern" tool.

    c.   Choose the center point for the pattern and specify the total angle or number of instances.

    d.   Adjust any additional settings, such as orientation or flipping options.

    e.   Click "OK" to create the circular pattern.

Rectangular Pattern:

The rectangular pattern creates a rectangular array of sketch entities or features by specifying the number of instances in each direction. Here's how to use the rectangular pattern:

a. Select the sketch entities or features you want to pattern.

b. Go to the toolbar and click on the "Rectangular Pattern" tool.

c. Choose the direction and specify the number of instances in each direction.

d. Adjust any additional settings, such as orientation or flipping options.

e. Click "OK" to create the rectangular pattern.

Sketch-Driven Pattern:

The sketch-driven pattern allows you to create a pattern using a sketch as a driving feature. Here's how to use the sketch-driven pattern:

a. Create a sketch that represents the pattern you want to generate.

b. Select the sketch entities you want to pattern.

c. Go to the toolbar and click on the "Sketch-Driven Pattern" tool.

d. Select the sketch that will drive the pattern.

e. Adjust any additional settings, such as orientation or flipping options.

f. Click "OK" to create the sketch-driven pattern.

Pattern Options:

In addition to the basic pattern types, Onshape provides various options to customize the pattern. These options include

a. Pattern Instances: You can define the number of instances or spacing for the pattern.

b. Direction: You can choose the direction of the pattern, such as linear or circular.

c. Flip Instances: You can flip instances to create mirror images within the pattern.

d. Orientation: You can control the orientation of the pattern instances.

e. Feature Scope: You can specify whether the pattern should include only sketch entities or extend to features.

Mirroring in Onshape

The Mirror tool in Onshape allows you to create mirrored copies of sketch entities, features, or bodies. It is useful for creating symmetrical designs and maintaining consistency across a model.

1. *Selecting Entities to Mirror:*

   Before using the Mirror tool, select the entities you want to mirror. This can include sketch entities, features, or bodies.

2. *Accessing the Mirror Tool:*

   Once the entities are selected, you can access the Mirror tool in one of the following ways:

   a. Toolbar: Go to the toolbar and click on the "Mirror" tool.
   b. Right-Click Menu: Right-click on the selected entities and choose the "Mirror" option.

3. *Defining the Mirror Plane:*

   After accessing the Mirror tool, you need to define the mirror plane. The mirror plane is the plane that acts as the reflection plane for the mirrored entities. Here's how to define the mirror plane:

   a. Select a plane: Choose an existing plane in the model or a reference plane as the mirror plane.
   b. Create a mirror plane: If the desired mirror plane doesn't exist, you can create a new reference plane to serve as the mirror plane.

4. *Adjusting the Mirror Settings:*

   Once the mirror plane is defined, you can adjust the mirror settings based on your requirements. Here are some common settings you can modify:

   a. Mirror Scope: You can choose to mirror only the selected entities or extend the mirror operation to additional entities or features.

b. Mirror Method: You can choose between "Feature" and "Body" methods. The "Feature" method mirrors individual features, while the "Body" method mirrors the entire body.

c. Mirror Options: Depending on the selected entities, you may have additional options specific to sketch entities, features, or bodies.

5. *Confirming the Mirror Operation:*

After adjusting the mirror settings, confirm the mirror operation by clicking "OK" or a similar confirmation button.

6. *Editing the Mirrored Entities:*

Once the mirror operation is completed, the mirrored entities will be created as separate instances. You can edit these instances individually or make changes to the original entities, which will be automatically reflected in the mirrored copies.

7. *Updating the Mirror:*

If you make changes to the original entities, such as modifying the sketch or feature, the mirror operation will update accordingly to maintain the mirrored symmetry.

Summary

There are many different ways to construct the same model. Onshape provides a variety of tools to allow you to build complex parts by taking advantage of symmetries and simple shapes. A list of Onshape features that we discussed in this curriculum guide has been provided below:

- Basic functions of sketching
- Planes/reference geometry
- Types of sketch tools
- Types of sketch lines
- Dimensioning
- Patterns
- Mirroring

Additional Resources

Onshape YouTube Video Tutorials:
- [Onshape Interface](#)
- [Planes](#)
- [Constraints](#)

Onshape Video Courses:
- [Navigating Onshape](#)
  (~35 min)
- [Intro to Sketching](#)
  (~33 min)

Onshape Basics Web Guide
- [Sketch Basics](#)