Design Intent

ENGR 1182
SolidWorks 4
Today’s Objectives

- Design Intent
  - Fully Defined
  - Design Analysis

- SW04 In-Class Activity
  - Fully Defining a Profile
  - Starbucks Coffee Cup Analysis

- SW04 Out-of-Class Homework Assignment
Design Intent

• Fully Defined Sketches
  • Create sketches that are reusable and robust

• Design Analysis
  • Determine the accuracy of designs relative to mass properties.
Fully Defined Sketches

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Fully Defined

- Full use of geometric and dimensional constraints
- Any additional information would be over-constraining
- There are multiple methods to end at the same goal
Anchoring a Sketch

- A Sketch must not only be fully constrained to the correct dimensions but also must be anchored to the origin.

- Well thought out designs will use the origin as a logical starting place of the design (such as the center point of a circle or square).
Fully Defined allows for Reusable Sketches

- Maintains shape and profile during adjustments to dimensions
- Especially needed for designs that are still evolving and need multiple versions
- Remember: Textbook problems are practice for students to learn from. In real life designers do not know the end goal before they begin.
SolidWorks Fully Defined

- In 2D sketches, a profile is completely defined by geometric and dimensional constraints.
- No “blue” lines or points
- Bottom right status bar displays “Fully Defined”
SolidWorks Fully Defined

- A strategy to maintain **Fully Defined** sketches which are constructed so that the **complexity is reduced** as much as possible:

1. After drawing a “closed” sketch, anchor it to the origin or some other point. Add any **geometric** constraints and **dimension** its various elements so that is **Fully Defined**.

2. If the object has symmetry, strongly consider **using symmetry constraints** to the origin or some constrained construction center line.

3. Use **equal constraints where appropriate** to reduce dimensioning complexity.

4. Draw another feature on the sketch and repeat steps 1-3.
Easily Select the Surface to be “Sketched” using HOT KEYS (See content on website)

(often used keys are circled)

### SolidWorks Hotkeys

<table>
<thead>
<tr>
<th>General</th>
<th>Change View</th>
<th>Zoom, Pan and Rotate</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ctrl + S</td>
<td>Ctrl + 1</td>
<td>Z</td>
</tr>
<tr>
<td>Ctrl + C</td>
<td>Ctrl + 2</td>
<td>Shift + Z</td>
</tr>
<tr>
<td>Ctrl + V</td>
<td>Ctrl + 3</td>
<td>F</td>
</tr>
<tr>
<td>Ctrl + X</td>
<td>Ctrl + 4</td>
<td>G</td>
</tr>
<tr>
<td>Ctrl + Y</td>
<td>Ctrl + 5</td>
<td>Ctrl + Arrows</td>
</tr>
<tr>
<td>Ctrl + Z</td>
<td>Ctrl + 6</td>
<td>Arrow Keys</td>
</tr>
<tr>
<td></td>
<td>Ctrl + 7</td>
<td>Shift + Arrows</td>
</tr>
<tr>
<td></td>
<td>Ctrl + 8</td>
<td>Alt + L/R Arrows</td>
</tr>
<tr>
<td></td>
<td>Ctrl + Shift + Z</td>
<td>Previous View</td>
</tr>
<tr>
<td></td>
<td>Spacebar</td>
<td>Views Toolbar</td>
</tr>
<tr>
<td></td>
<td>Ctrl + Spacebar</td>
<td>Selector Cube</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Zoom Out</td>
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<tr>
<td></td>
<td></td>
<td>Zoom In</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Zoom to Fit</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Magnify</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Pan</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Rotate 15°</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Rotate 90°</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Rotate</td>
</tr>
</tbody>
</table>
Looking Ahead – on the Midterm you will be required to save your files as PDFs

1) Click on “File/ Save As”

2) Under “Save As Type:” click on “Adobe Portable Document Format(*.pdf)”
Fully Defined

Wrap Up

Fully Defined

- All Constraints Applied
  - Geometric
  - Dimensional
- Anchored Sketches
- Allows for Reusable Sketches

Homework Assignment SW04-OUT:
Problem 6.4 (aa)

See Out_Of_Class_HW_Help on EEIC website
Fully Define In-Class Problem

- Add Geometric and Dimensional Constraints to Fully Define the 2D sketch
- The left profile has been fully defined (all black) and can be used as a reference (dimensions) and as a guide (method)

Note: The 3.00" dimension from the y-axis is an arbitrary dimension in order to anchor the location of the sketch.
Design Analysis

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Design Analysis

- Mass Properties
  - Center of Mass
  - Volume
  - Surface Area
  - Moments of Inertia

Can be used to check the accuracy of the model. To assure a model will have the correct center of mass and volume all sketches must be “Fully Defined”
The Mass Properties Toolbox in SolidWorks is located in the Tools menu. This menu is found by clicking the arrow next to the SolidWorks logo and then finding Mass Properties under Tools.
SolidWorks Mass Properties Toolbox

The following dialog box will appear and display the Mass Properties of the Solid Model.
Design Analysis Wrap Up

- Mass Properties Toolbox
  - Volume
  - Surface Area
  - Center of Mass

Homework
Problem 8.11 (g)
In-Class Problem

Starbucks© Sizes

Model the Starbucks cups using the simple sketch below and a revolve. Fully define the sketch and reuse with each new dimension set. Obtain the Mass Properties requested for each size.

<table>
<thead>
<tr>
<th>Starbucks Sizes</th>
<th>Small Diameter (in)</th>
<th>Big Diameter (in)</th>
<th>Height (in)</th>
<th>Volume (in^3)</th>
<th>Volume (oz.)</th>
<th>Surface Area (in^3)</th>
<th>Center of Mass (in) - from bottom of cup</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tall</td>
<td>2.5</td>
<td>3.5</td>
<td>3.75</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Grande</td>
<td>2.25</td>
<td>3.5</td>
<td>5</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Venti</td>
<td>2.375</td>
<td>3.5</td>
<td>6</td>
<td></td>
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<td></td>
</tr>
</tbody>
</table>
Important Takeaways

- When solid modeling, designers need to take into account the design intent and make sure the sketches used are fully defined.
- Mass property tools in CAD are a useful tool to analyze a part after it is made.
What’s Next?

- Due Next Class: SW04 Out of Class HW
- Feature based modeling is breaking down a part into features to create the part in the most efficient way.
- Take SW05 Quiz on readings