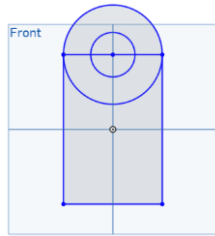


College Curriculum Outline

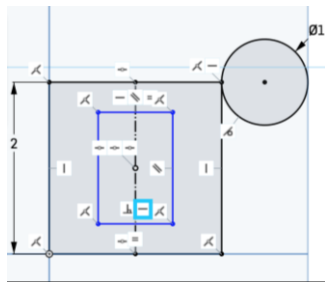
Welcome to the Onshape College Curriculum! This outline briefly explains what kind of topics will be covered every week:

Week 1: Getting Started



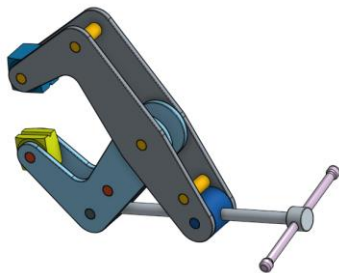
- Creating an Onshape account
- Navigating a 3D environment
- Explaining sketch-based modeling
- Introducing the 4 foundational features (extrude, revolve, sweep, and loft)
- Transitioning from 2D to 3D
- Introducing basic sketching

Week 2: Parts



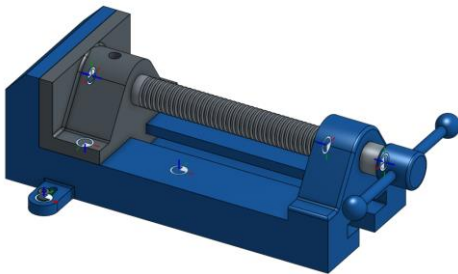
- An introduction to “Design Intent”
- Using dimensions and constraints
- Automatic inferencing
- Making an accurate part
- Sketching practice
- Using and creating planes
- Creating fillets and chamfers
- Utilizing multiple sketch regions
- Basic parts

Week 3: Multi-Part Part Studio



- Using Boolean operations
- Applying linear and circular patterning
- An introduction to concurrent top-down and bottom-up designs
- Creating a Multi-Part design in a Part Studio

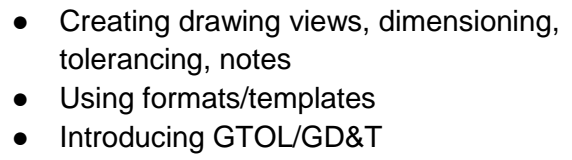
Week 4: Assemblies



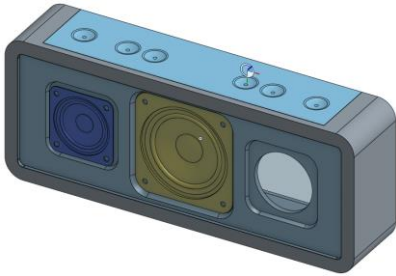
- Lesson on degrees of freedom
- An introduction to assembly Mates
- Mate Connectors
- Manipulating part position with the triad
- Explaining Mates and Relations
- Animating Mates
- An introduction to Linked Documents
- Applying limits to a Mate

Week 5: 2D Drawings

- An introduction to engineering drawings
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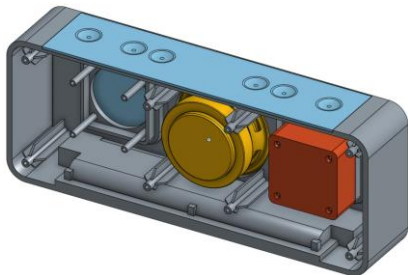


Week 6: Product Design Within Teams



- Starting Bluetooth Speaker project
- Creating teams
- Using derived parts
- Organizing the Feature Tree
- Simultaneous collaboration (i.e. "Google Docs-style collaboration")
- Including comments
- Following

Week 7: Iterative Design



- Continuing Bluetooth Speaker project
- Using FeatureScript for screw bosses and ribs
- Adding additional model detail
- Version control and history
- Re-ordering parametric features
- Exercising top-down design

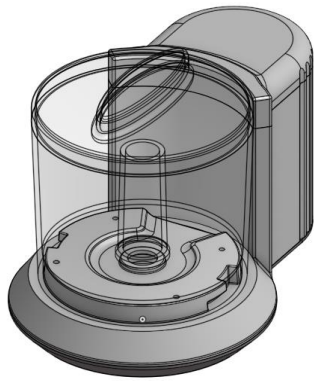
Week 8: Advanced Assembly



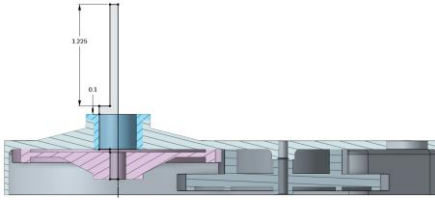
- Continuing Bluetooth Speaker project
- Using Linked Documents for standard hardware
- Advanced Assembly concepts
- Applying "snap mode" in Assembly
- Grouping in Assembly
- Replicating for fasteners

Week 9: Advanced Geometry & Design

- Starting Chopper project
 - Advanced part modeling
 - Advanced top-down design
 - Applying drafts
 - Using surfaces
 - Splitting parts
 - Using variables/expressions
 - Editing appearance/transparency
-



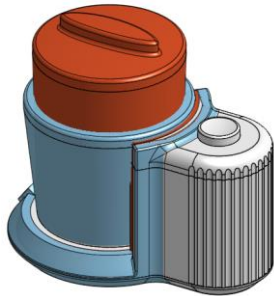
Week 10: Design for Manufacturing



- Using the Hole Tool
- Using FeatureScript for spur gears
- Importing Solidworks® CAD Pack/Go files
- Direct editing an existing part (modify fillet, delete/move/replace face)
- An introduction to the Onshape App Store (through a look at a CAM app)

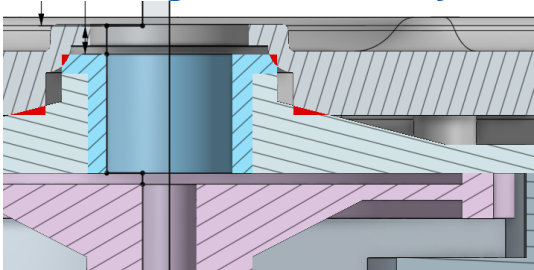
Week 11: Product Data Management

Design 2 Design 1



- Advanced part modeling
- Lofting
- Importing and manipulating sketch picture
- Sketching with splines
- Embossing logo
- Drawing a helix to make a spring
- Using Branch/Compare/Merge features

Week 12: Advanced Tools & Design for Assembly



- Using section view to look for interference
 - Applying Gear Relations
 - Applying materials and using Mass Properties
 - Using the explode view from App Store
 - Making edits from Onshape Mobile
 - Exporting Solidworks® CAD files
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