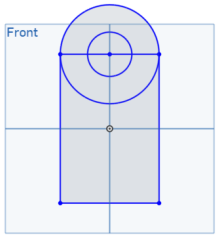
Onshape College Curriculum Outline

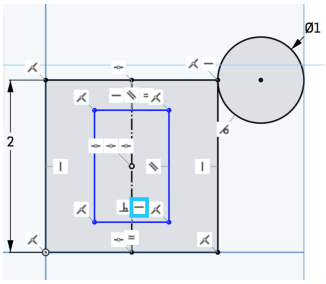
Welcome to the Onshape College Curriculum! This outline briefly explains what kind of topics will be covered in every lesson:

Lesson 1: Getting Started



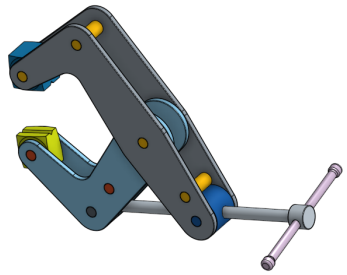
- Creating an Onshape account
- Navigating a 3D environment
- Explaining sketch-based modeling
- Introducing the 4 foundational features (extrude, revolve, sweep, and loft)
- Transitioning from 2D to 3D
- Introducing basic sketching

Lesson 2: Parts



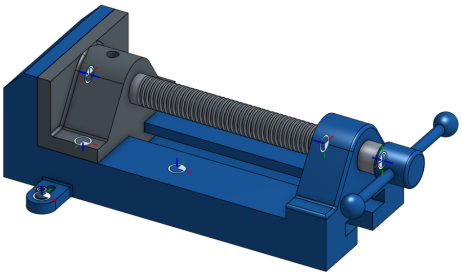
- An introduction to “Design Intent”
- Using dimensions and constraints
- Automatic inferencing
- Making an accurate part
- Sketching practice
- Using and creating planes
- Creating fillets and chamfers
- Utilizing multiple sketch regions
- Basic parts

Lesson 3: Multi-Part Part Studio



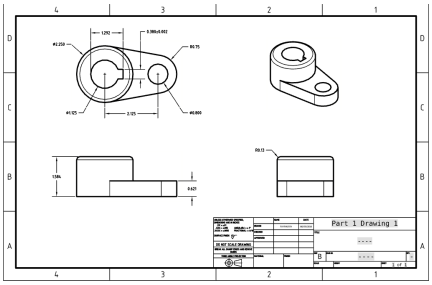
- Using Boolean operations
- Applying linear and circular patterning
- An introduction to concurrent top-down and bottom-up designs
- Creating a Multi-Part design in a Part Studio

Lesson 4: Assemblies



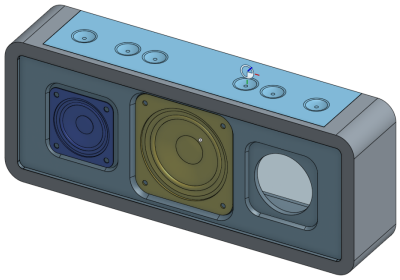
- Lesson on degrees of freedom
- An introduction to assembly Mates
- Mate Connectors
- Manipulating part position with the triad
- Explaining Mates and Relations
- Animating Mates
- An introduction to Linked Documents
- Applying limits to a Mate

Lesson 5: 2D Drawings



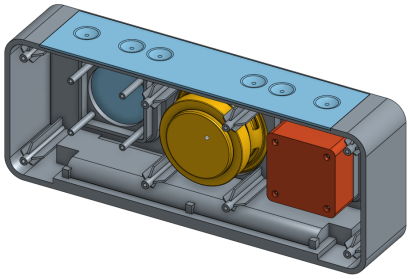
- An introduction to engineering drawings
- Creating drawing views, dimensioning, tolerancing, notes
- Using formats/templates
- Introducing GTOL/GD&T

Lesson 6: Product Design Within Teams



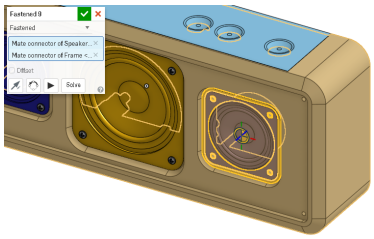
- Starting Bluetooth Speaker project
- Creating teams
- Using derived parts
- Organizing the Feature Tree
- Simultaneous collaboration (i.e. “Google Docs-style collaboration)
- Including comments
- Following

Lesson 7: Iterative Design



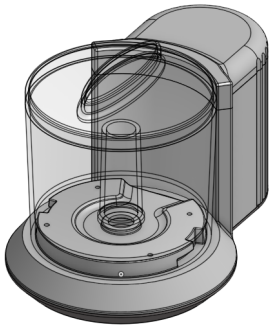
- Continuing Bluetooth Speaker project
- Using FeatureScript for screw bosses and ribs
- Adding additional model detail
- Version control and history
- Re-ordering parametric features
- Exercising top-down design

Lesson 8: Advanced Assembly



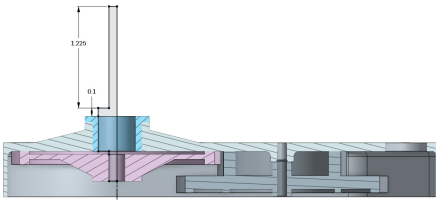
- Continuing Bluetooth Speaker project
- Using Linked Documents for standard hardware
- Advanced Assembly concepts
- Applying “snap mode” in Assembly
- Grouping in Assembly
- Replicating for fasteners

Lesson 9: Advanced Geometry & Design



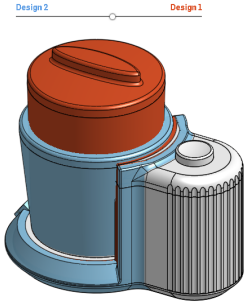
- Starting Chopper project
- Advanced part modeling
- Advanced top-down design
- Applying drafts
- Using surfaces
- Splitting parts
- Using variables/expressions
- Editing appearance/transparency

Lesson 10: Design for Manufacturing



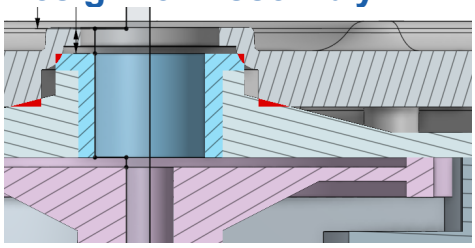
- Using the Hole Tool
- Using FeatureScript for spur gears
- Importing Solidworks® CAD Pack/Go files
- Direct editing an existing part (modify fillet, delete/move/replace face)
- An introduction to the Onshape App Store (through a look at a CAM app)

Lesson 11: Product Data Management



- 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

Lesson 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