

# Onshape beginner kata\*

## The practice of basic Onshape techniques

18<sup>th</sup> December 2020

**David Ott**

[David.Ott@unige.ch](mailto:David.Ott@unige.ch)

**FacLab of University of Geneva**

[FacLab@unige.ch](mailto:FacLab@unige.ch)



\* In Japanese language kata (though written as 方) is a frequently-used suffix meaning “way of doing,” with emphasis on the form and order of the process. Other meanings are “training method” and “formal exercise.” The goal of a painter's practising, for example, is to merge their consciousness with their brush; the potter's with their clay; the garden designer's with the materials of the garden. Once such mastery is achieved, the theory goes, the doing of a thing perfectly is as easy as thinking it.

More info: <https://en.wikipedia.org/wiki/Kata>

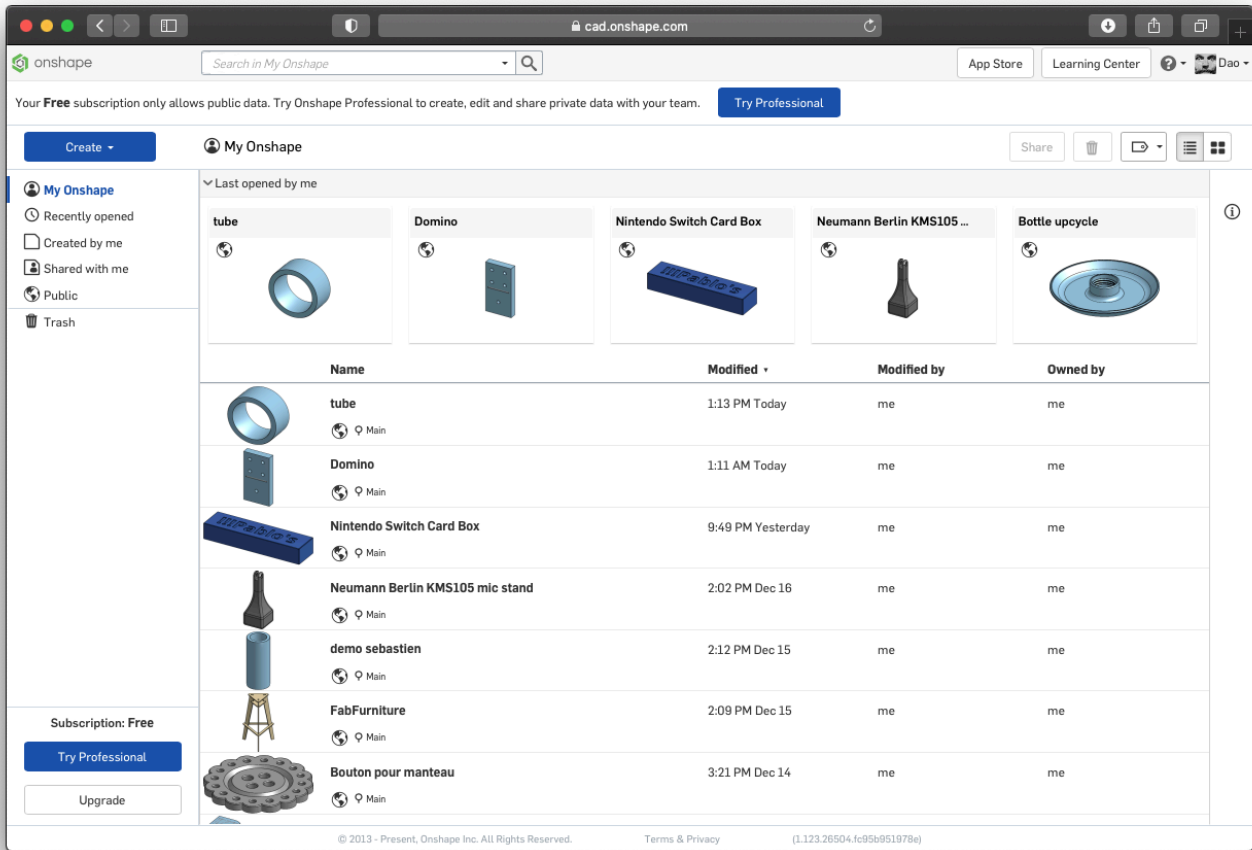
# Table of contents

Table of contents	2
<b>Getting started</b>	<b>4</b>
Create a document	4
The “basic interface” kata	5
Topics & techniques	5
The “workspace units” kata	6
Change length unit format to millimeters	6
Warm-up	7
Make a cube with an extrusion	7
Make a hole in the cube with an extrusion	7
Make a cylinder with an extrusion	7
Make a cake with successive extrusions	7
Make a cylinder with a revolution	7
<b>The “extrusion tube” kata</b>	<b>7</b>
Create a document	8
Sketch 2 circles	8
Extrude a ring to form a tube	8
Variables definition	8
Dimensioning the tube with the define variables	8
The “2D export” kata	9
Export a face (in 2D) of an object as an .DXF	9
The “3D export” kata	10
Export an object (in 3D) as an .STL (or .OBJ)	10
The “advanced interface” kata	11
Topics & techniques	11
The “Revolve tube” kata	12
Create a new Part Studio (using Duplicate)	12
Delete some features	12
Sketch a rectangle + construction line	12
Revolve a rectangle around a construction line	12
Additional variables definition	12
Dimensioning the tube with the defined variables	12
The “view keyboard shortcuts” kata	14
The “bead” kata	15
Create a new document	15
Sketch half a circle	15
Revolve tool to make a sphere	15
Extrude a cylinder to prepare a hole tool	15
Boolean tool to make a hole in the bead	15
The “watchface” kata	16
Create a new document	16
Sketch-a-watchface	16
Sketch 1 cont’d: Constrain circles to origin	16
Sketch 1 cont’d: Offset tool	16
Sketch 1 cont’d: Construction tool	16
Sketch 1 cont’d: Midpoint & coincident	16
Sketch 1 cont’d: Circular pattern	17
Extrude watchface profile	17
The “better bead” kata	18
Open existing document and duplicate a tab	18
Delete features	18
Sketch a better bead	18
Revolve a better bead	18
Chamfer & fillet	18

# Getting started

## Create a document

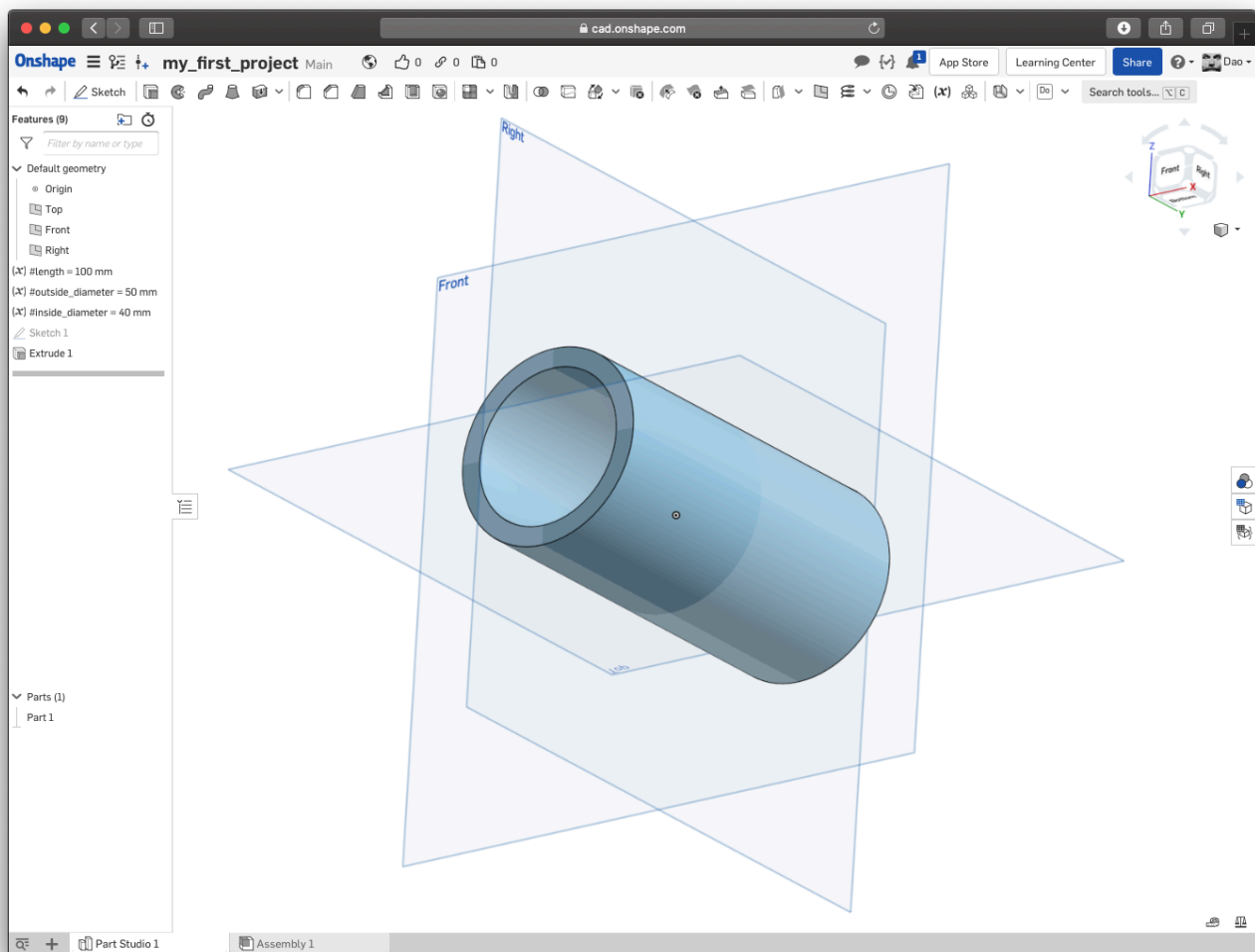
→ Create > Document...



# The “basic interface” kata

## Topics & techniques

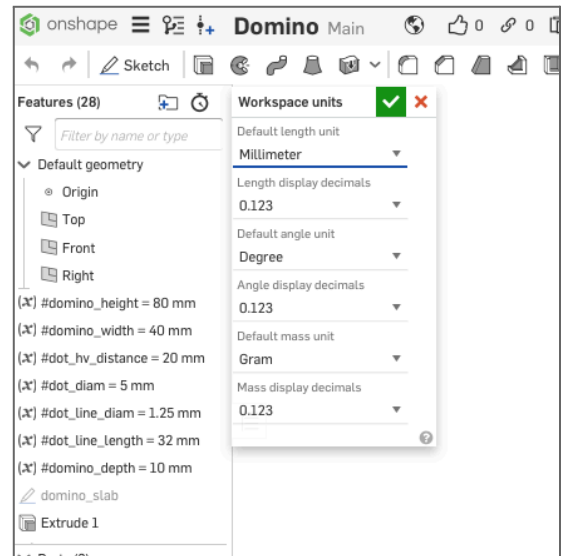
- Toolbar
- Tab bar
- Features list
- Navigation
  - ◆ Zoom
  - ◆ Pan
  - ◆ 3D “cube”



# The “workspace units” kata

## Change length unit format to millimeters

- Click on the **hamburger shaped icon** next to the top left “onshape” logo
- Select the **Workspace units...** option
- If required
  - ◆ Change **Default length unit** to **millimeters**
  - ◆ Change any required other units
- Validate **Workspace units**



# Warm-up

Make a cube with an extrusion

Make a hole in the cube with an extrusion

Make a cylinder with an extrusion

Make a cake with successive extrusions

Make a cylinder with a revolution

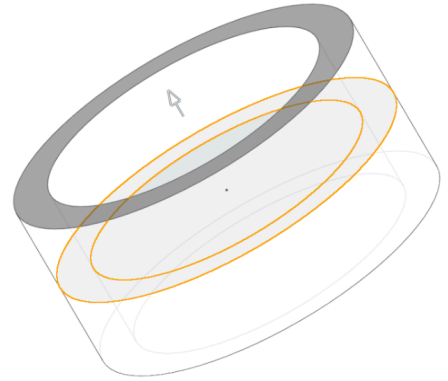
# The “extrusion tube” kata

## Create a document

- Name = Tube

## Sketch 2 circles

- Click **sketch** tool (or shift-s)
  - ◆ Click **Sketch plane**
    - Click on **Front** plane
  - ◆ Click **center point circle**
    - Draw a circle centered on origin
    - Draw another **center point circle** inside the first
  - ◆ Validate **Sketch 1**



## Extrude a ring to form a tube

- Click **extrude** tool (or shift-e)
  - ◆ Click **Faces and sketch regions to extrude**
    - Click on the outer ring of Sketch 1
    - Note the default Depth value = 25mm
    - Change **End Type** to **Symmetric**
  - ◆ Validate **Extrude 1**

## Variables definition

- Click **variable** tool
  - ◆ Name = outer\_diameter
  - ◆ Value = 50mm
  - ◆ Validate variable **#outer\_diameter**
- Click **variable** tool
  - ◆ Name = inner\_diameter
  - ◆ Value = 40mm
  - ◆ Validate variable **#inner\_diameter**
- Click **variable** tool
  - ◆ Name = tube\_length
  - ◆ Value = 100mm
  - ◆ Validate variable **#tube\_length**
- Select all variables
- Move all variables to top of features list (before **Sketch 1**)
- While all variables are still selected click on **New folder** and name it **vars**
- Expand **vars** folder to see all your variables

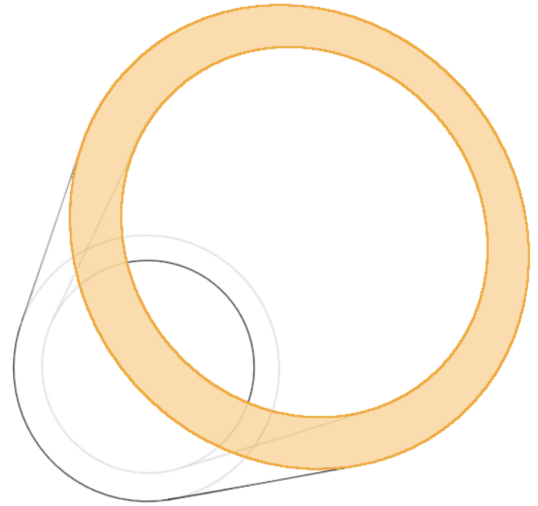
## Dimensioning the tube with the define variables

- Double-click **Sketch 1** feature
  - ◆ Click **dimension** tool
    - Select the outer circle
    - Value = **#outer\_diameter**
    - Validate the dimension with enter key
  - ◆ With **dimension** tool still selected
    - Select the inner circle
    - Value = **#inner\_diameter**
    - Validate the dimension with enter key
  - ◆ Validate **Sketch 1**
- Edit **Extrude 1** feature
  - ◆ Assign **#tube\_length** to **Depth**
  - ◆ Validate **Extrude 1**

# The “2D export” kata

Export a face (in 2D) of an object as an .DXF

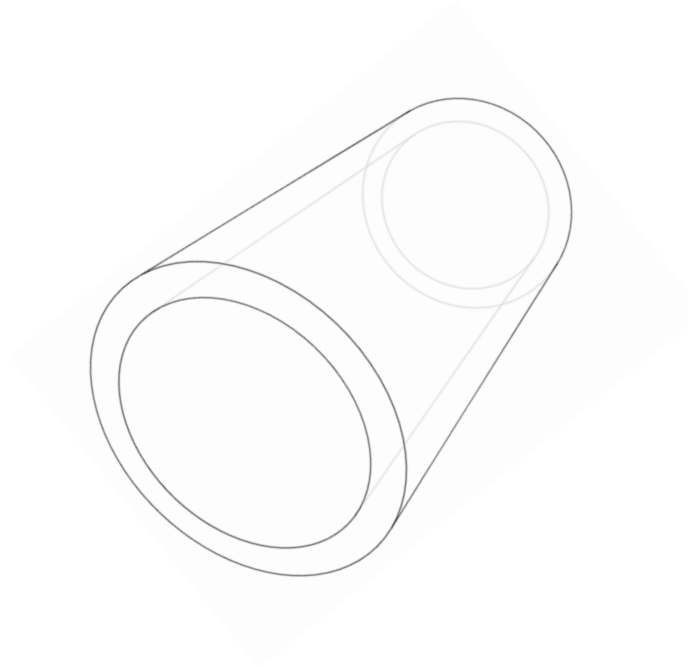
- Right-click on face to export
  - ◆ Select **Export as DXF/DWG** from menu
  - ◆ Select **Format = DXF**
  - ◆ Validate export with **OK**



# The “3D export” kata

Export an object (in 3D) as an .STL (or .OBJ)

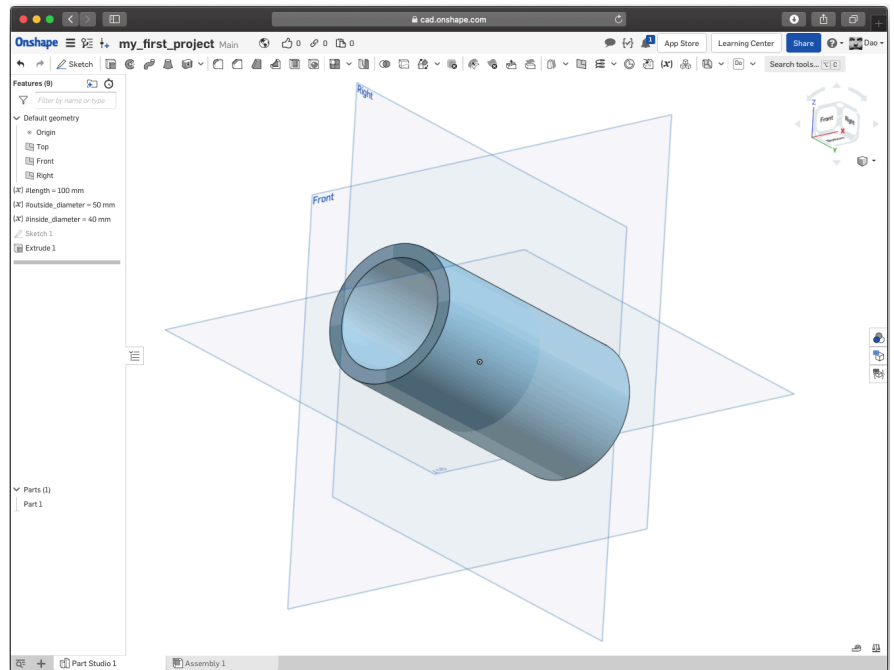
- Right-click on part (i.e. object) to export
  - ◆ Select **Export...** from menu
  - ◆ Select **Format = STL (or OBJ)**
  - ◆ Validate export with **OK**



# The “advanced interface” kata

## Topics & techniques

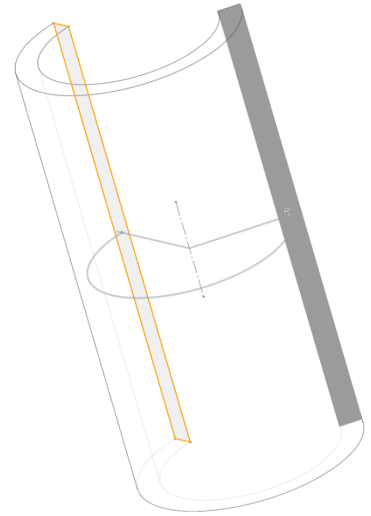
- Part list
- Camera and render options
- Sharing
- Selection
  - ◆ Click to select / unselect
  - ◆ Space to unselect all



# The “Revolve tube” kata

## Create a new Part Studio (using Duplicate)

- Open **Tube** document
- Right click on the tab **Part Studio 1** and rename to **Extrude Tube**
- Right click on the tab **Extrude Tube** and duplicate
- Right click on the tab **Extrude Tube Copy 1** and rename to **Revolve Tube**



## Delete some features

- Click **Extrude 1**
- Click **Sketch 1**
- Delete both features

## Sketch a rectangle + construction line

- Click **sketch** tool (or “shift-s”)
  - ◆ Click **Sketch plane**
    - Click on the **Front** plane
  - ◆ Right click somewhere and select **View normal to sketch plane**
    - Or click on Front on the navigation “3D cube”
  - ◆ Click **Center point rectangle** (or “r” key)
    - Move your cursor over the origin (to activate guide line) then slightly to the left of it
    - Click and draw a rectangle vertically centered on origin but a little to the left of it
  - ◆ Click **Line** (or “l” key) and **Construction** (or “q” key)
    - Draw a vertical construction line that passes through the origin
    - Hit the **escape key** (“esc” key) to exit the line drawing mode
  - ◆ Validate **Sketch 1**

## Revolve a rectangle around a construction line

- Click **Revolve** tool
  - ◆ Click **Faces and sketch regions to extrude**
    - Click on the rectangle (in Sketch 1)
  - ◆ Click on **Revolve axis**
    - Select the vertical construction (in Sketch 1)
  - ◆ Validate **Revolve 1**

## Additional variables definition

- Open **vars** folder in the feature list
- Click **variable** tool
  - ◆ Name = tube\_thickness
  - ◆ Value =  $(\#outer\_diameter - \#inner\_diameter) / 2$
  - ◆ Validate variable **#tube\_thickness**
- Click **variable** tool
  - ◆ Name = tube\_radius
  - ◆ Value =  $\#outer\_diameter / 2$
  - ◆ Validate variable **#tube\_radius**
- Select **#tube\_thickness** and move into **vars** folder

## Dimensioning the tube with the defined variables

- Double-click **Sketch 1** feature
  - ◆ Click the **dimension** tool
    - Select the horizontal edge of the rectangle
    - Value = **#tube\_thickness**
    - Validate the dimension with enter key
  - ◆ With the **dimension** tool still selected

- Select the vertical edge of the rectangle
- Value = **#tube\_length**
- Validate the dimension with enter key
- ◆ With the **dimension** tool still selected
  - Select both the origin and the distal vertical edge of the rectangle
  - Value = **#outer\_diameter**
  - Validate the dimension with enter key
- ◆ Validate **Sketch 1**

# The “view keyboard shortcuts” kata

- Front view = **Shift 1**
- Back view = **Shift 2**
- Left view = **Shift 3**
- Right view = **Shift 4**
- Top view = **Shift 5**
- Bottom view = **Shift 6**
- Isometric view = **Shift 7**
- Section view = **Shift X**
- Named view = **Shift V**

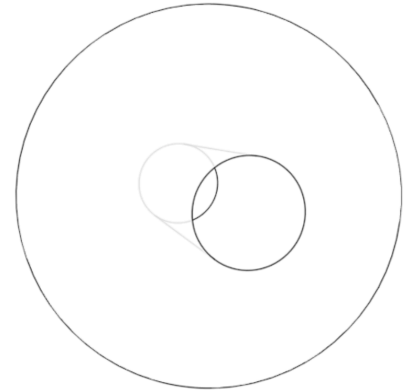
# The “bead” kata

## Create a new document

- OnShape > Create > Document > Name it = “Bead”

## Sketch half a circle

- Click on Sketch tool
  - ◆ Select Sketch plane = Front
  - ◆ Draw a center point circle (centered on origin)
  - ◆ Assign a dimension to the circle = 20mm
  - ◆ Draw a vertical line cutting the circle through its center
    - Note the constraints inferences when you move the mouse pointer close to the circle and aligned with the vertical from the circle center
  - ◆ Press escape to stop drawing lines
  - ◆ Validate Sketch 1



## Revolve tool to make a sphere

- Click on Revolve tool
- Select Faces and sketch regions to revolve
- Click on one half of the circle in Sketch 1
- Select Revolve axis
- Click on vertical line in Sketch 1
- Validate Revolve 1

## Extrude a cylinder to prepare a hole tool

- Hide Part 1
- Click on Sketch tool to create a Sketch 2
  - ◆ Select Sketch plane = Top
  - ◆ Draw another Center point circle (centered on origin)
  - ◆ Assign a dimension to the circle = 5mm
  - ◆ Validate Sketch 2
- Click on Extrude tool
  - ◆ Select Faces and sketch regions to extrude
  - ◆ Click on the circle in Sketch 2
  - ◆ Select Symmetric option (instead of Blind)
  - ◆ Assign a depth = 40mm
  - ◆ Validate Extrude 1
- Show Part 1

## Boolean tool to make a hole in the bead

- Click on Boolean tool
  - ◆ Select Subtract (instead of Union)
  - ◆ Select Tools
  - ◆ Click on the cylinder extrusion
  - ◆ Select Targets
  - ◆ Click on the sphere (the result of the revolve feature)
  - ◆ Validate Boolean 1

# The “watchface” kata

## Create a new document

- OnShape > Create > Document > Name it = “Watchface”

## Sketch-a-watchface

- Click Sketch tool to create Sketch 1
  - ◆ Sketch plane = Front
  - ◆ Draw a Center point circle somewhere
  - ◆ Type “100” then hit return
    - This directly assigns the value 100mm to the circle diameter
  - ◆ Draw another circle with diameter 15mm (somewhere else)
  - ◆ Draw another circle with diameter 5mm (yet somewhere else)
  - ◆ (Sketch 1 to be continued on next slide)



## Sketch 1 cont'd: Constrain circles to origin

- ◆ (still editing Sketch 1)
- ◆ For each circle
  - Click on the Concentric constraint
  - Select the circle center and the origin
    - Each circle will be constrained concentrically to the origin
- ◆ (Sketch 1 to be continued on next slide)

## Sketch 1 cont'd: Offset tool

- (still editing Sketch 1)
- Click Offset tool
- Select the big (100mm) circle
  - ◆ Verify that the offset value = 5mm
  - ◆ You can modify this value if you desire!
- (Sketch 1 to be continued on next slide)

## Sketch 1 cont'd: Construction tool

- ◆ (still editing Sketch 1)
- ◆ Click Construction tool
- ◆ Draw a vertical line cutting through all circles and origin
  - Note the construction line is of type dot-line-dot-line-etc.
- ◆ (Sketch 1 to be continued on next slide)

## Sketch 1 cont'd: Midpoint & coincident

- ◆ (still editing Sketch 1)
- ◆ Draw a rectangle 5mm wide and 15mm high
- ◆ Click Construction tool and click Point tool
- ◆ Place the point somewhere on the bottom edge of the rectangle
- ◆ Click Midpoint constraint
- ◆ Select the point and the bottom edge of the rectangle
- ◆ Click Coincident constraint
- ◆ Select the point and the vertical construction line
- ◆ Click Dimension tool
- ◆ Select 100mm circle and the top edge of the rectangle and space by 5mm
- ◆ (Sketch 1 to be continued on next slide)

## Sketch 1 cont'd: Circular pattern

- ◆ (still editing Sketch 1)
- ◆ Click Circular pattern tool
- ◆ Select (click and drag selection) all 4 points of the rectangle
- ◆ Change pattern value = 12
  - Note the circular pattern center has been placed by default on the origin, but this can be changed
- ◆ Double click to validate the circular pattern
- ◆ Validate Sketch 1

### Extrude base of watchface

- Create variable "profile" with value = 2mm
- Click Extrude tool to create "Extrude 1"
- Select all of Sketch 1 (click and drag to select all)
  - ◆ Unselect the small circle, this will create a center hole
- Select opposite direction
- Assign variable #profile to Depth value
- Validate Extrude 1

## Extrude watchface profile

- Click Extrude tool to create Extrude 2
- Show Sketch 1
  - ◆ Note that it was automatically hidden when validating "Extrude 1"
- Select outer ring and inner ring (around the center hole)
- Select all 12 marks
- Assign variable #profile to Depth value
- Validate Extrude 2

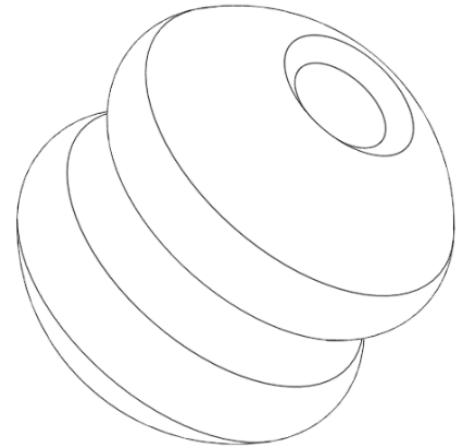
# The “better bead” kata

## Open existing document and duplicate a tab

- OnShape > Open document > Bead
- Right click Studio Part 1 tab > Duplicate
- Right click Studio Part 1 Copy 1 > Rename > Better bead

## Delete features

- Select Boolean 1 > Delete
- Select Extrude 2 > Delete
- Select Sketch 2 > Delete
- Select Revolve 1 > Delete



## Sketch a better bead

- Edit Sketch 1
  - ◆ Draw a line somewhere
  - ◆ Click vertical constraint tool
  - ◆ Select the new line to make it vertical
  - ◆ Click parallel constraint tool
  - ◆ Select the new line and previous vertical line
  - ◆ Click dimension tool
  - ◆ Select both lines > drag to position value label > assign value = 2.5mm
  - ◆ Click center point circle tool
  - ◆ Draw a new circle with center on the left and on the line of existing circle coincident to the Top plane and assign diameter value = 5mm
- Validate Sketch 1

## Revolve a better bead

- Click on Revolve tool
  - ◆ Select Faces and sketch regions to extrude
  - ◆ Click on left half of the circle in Sketch 1
    - Make sure not to include the small vertical slice left of middle line and the small circle on then left of the circle
  - ◆ Select Revolve axis
  - ◆ Click on vertical line in Sketch 1
  - ◆ Validate Revolve 1

## Chamfer & fillet

- Click on chamfer tool
  - ◆ Select Entities to chamfer
  - ◆ Click on 2 small circle left on both north and south poles of the bead
  - ◆ Assign distance value = 1mm
  - ◆ Validate chamfer 1
- Click on fillet tool
  - ◆ Select Entities to fillet
  - ◆ Click on the 2 circular edges left by the revolve around the equator of the bead
  - ◆ Assign radius value = 3mm
  - ◆ Validate fillet 1