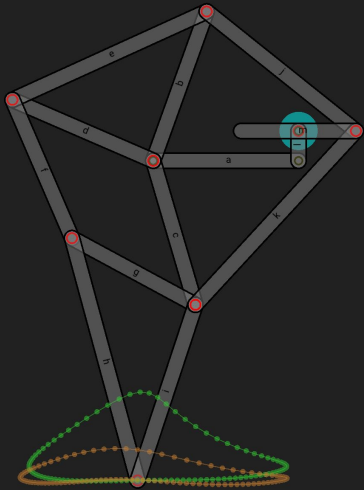


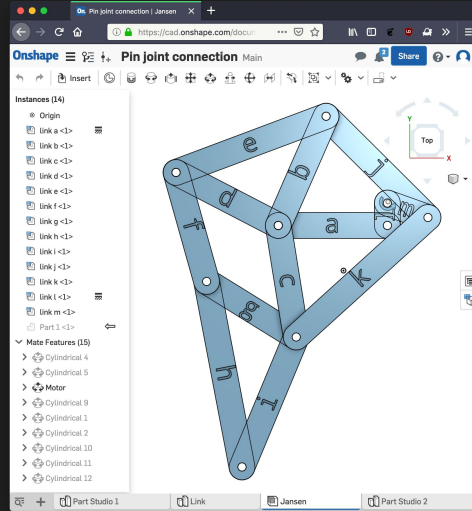
# Onshape & Fabrication

Quick Tutorial

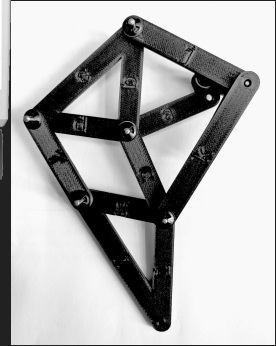
# From design optimization to fabrication



determine dimensions



modify final geometry



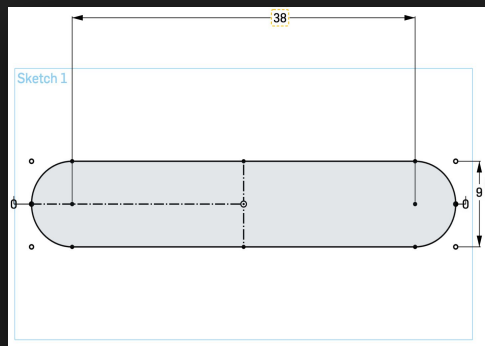
print and assemble!

*link lengths*

*triangle mesh (.stl)*

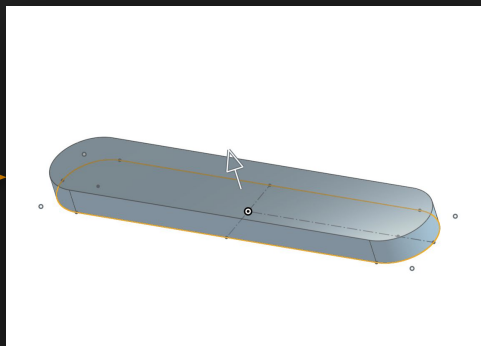
# Onshape - Creating a part

2D



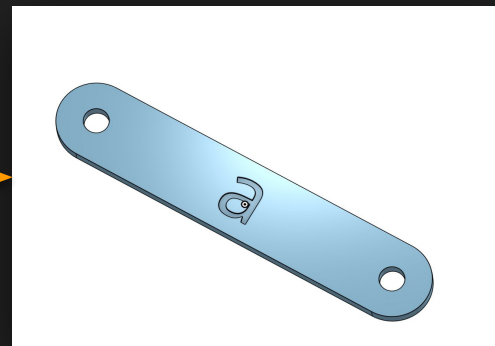
sketch, line

3D



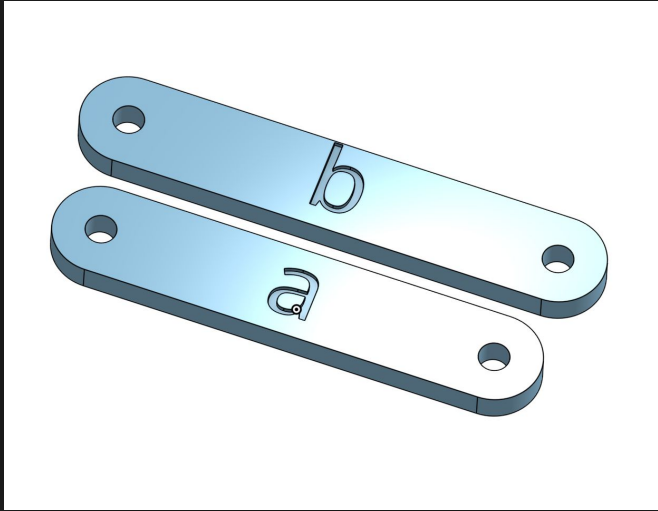
extrude, revolve, sweep,  
loft

modified 3D



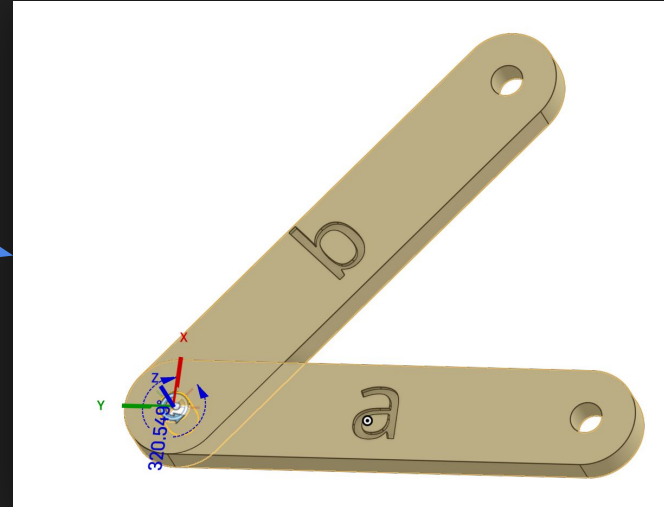
fillet, shell, hole,  
subtract-extrude, ...

# Onshape - Assemble parts



define relative placements  
with mates

mates == constraints  
→ Onshape runs a  
constrained rigid body  
simulation to position parts -  
just as you implemented!



# Onshape - Quick demo

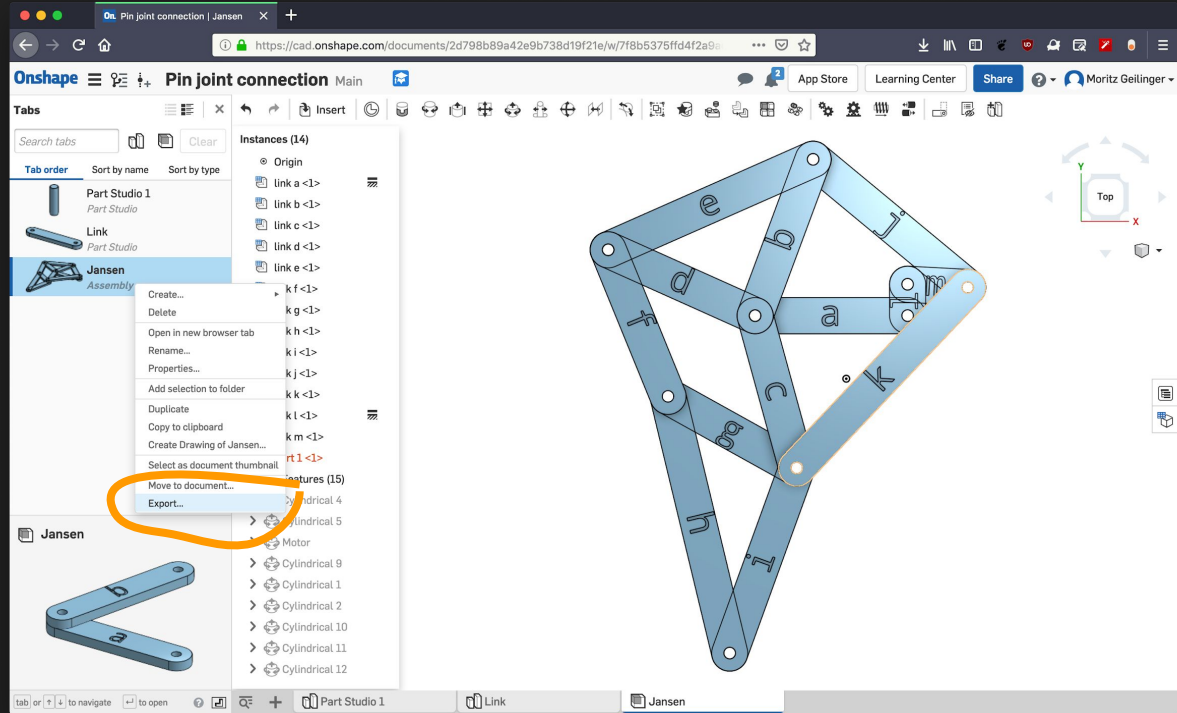
# Onshape - Configurations

The screenshot shows the Onshape web interface for a part named 'Link'. The 'Link' part is highlighted in the left sidebar. The main workspace shows a 3D model of the part. The 'Configurations' panel on the right displays a table of configurations. A blue circle highlights the 'Distance' column, and a purple circle highlights the 'Link f' row. A blue arrow points to the 'Distance' column with the text 'Enter dimensions'. A purple arrow points to the 'Link f' row with the text 'Open configurations tab'. An orange arrow points to the 'Link' part in the sidebar with the text 'Go to part "Link"'. The table lists configurations for 'Link a' through 'Link m' and 'Spacer', with dimensions in mm and scale factors.

Name	Distance	Text
Link a	30 mm * #scale	a
Link b	41.5 mm * #scale	b
Link c	39.3 mm * #scale	c
Link d	40.1 mm * #scale	d
Link e	55.8 mm * #scale	e
Link f	39.4 mm * #scale	f
Link g	36.7 mm * #scale	g
Link h	65.7 mm * #scale	h
Link i	49 mm * #scale	i
Link j	50 mm * #scale	j
Link k	61.9 mm * #scale	k
Link l	7.8 mm * #scale	l
Link m	30 mm * #scale / 2	m
Spacer		

# Onshape - Export geometries

right-click assembly  
→ Export

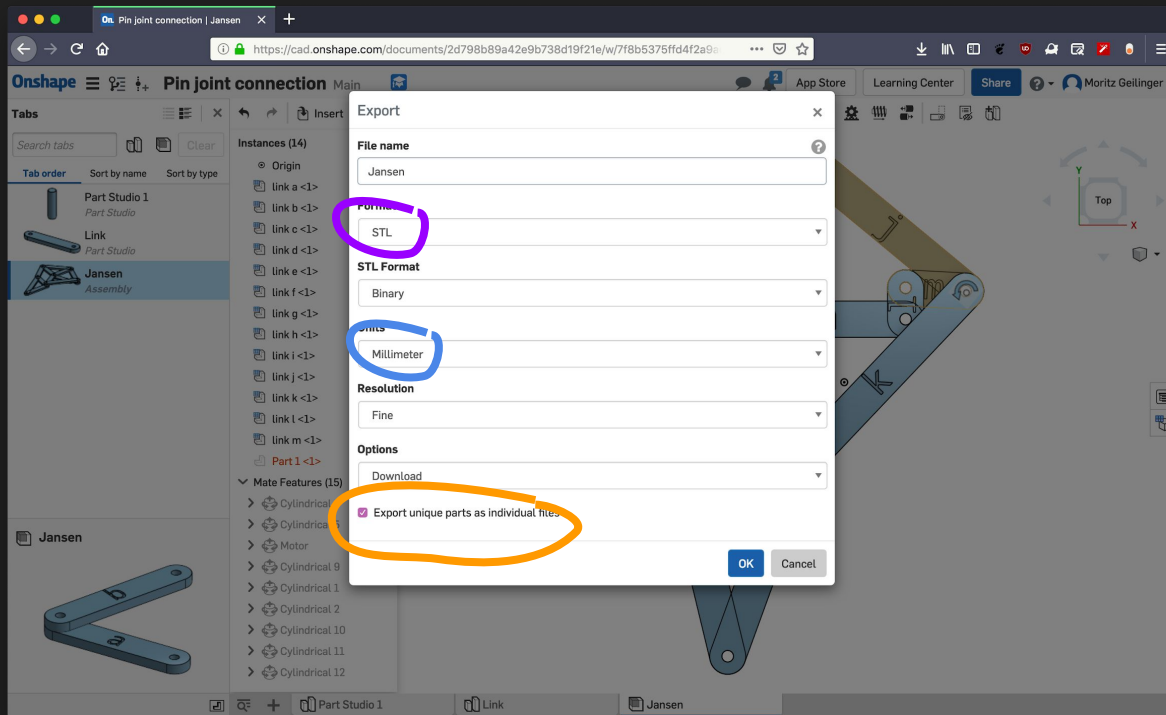


# Onshape - Export geometries

Choose **STL** as file format

Units: **millimeters**

Option to export as **unique files**





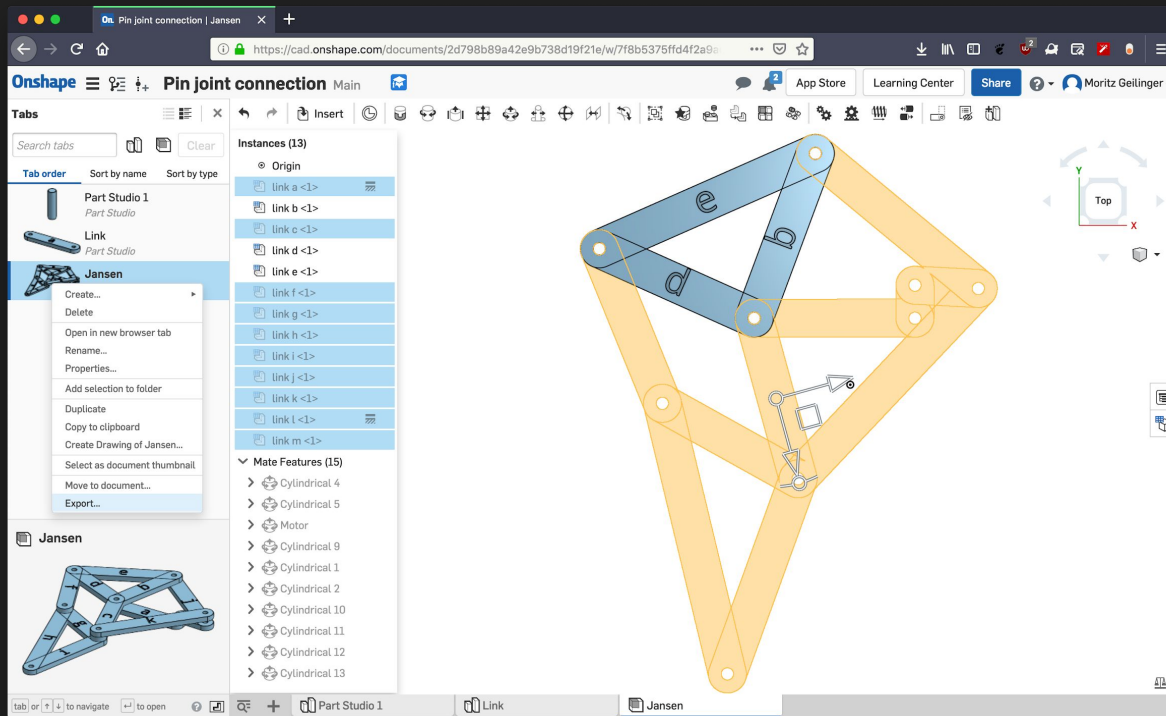
# Onshape - Export geometries: a small trick

Want to merge multiple links

Steps:

1. hide all other parts
2. right-click assembly  
→ export
3. Uncheck "Export as individual parts"

→ one STL with merged geometry



# Fabrication

Send a zip file containing all STLs to [moritzge@inf.ethz.ch](mailto:moritzge@inf.ethz.ch) by March 29 (next week)

You will get the printed parts on April 5, ready to be assembled!

Connections:

- Pins of length 10, 16, and 20mm



# Jansen Linkage template file

If you like, you can use the Jansen linkage template and adjust the dimensions in the configuration table of the part “Link”:

[https://cad.onshape.com/documents/2d798b89a42e9b738d19f21e/w/7f8b5375fd4f2a9aeaa79a9/e/18c64241d3b31a7050360cff?configuration=List\\_90owNnAf6up67c%3DSpacer](https://cad.onshape.com/documents/2d798b89a42e9b738d19f21e/w/7f8b5375fd4f2a9aeaa79a9/e/18c64241d3b31a7050360cff?configuration=List_90owNnAf6up67c%3DSpacer)