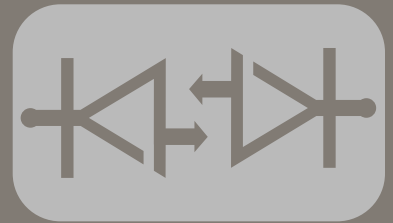


# FUSION360

*An introduction to cloud-based, parametric modeling for anyone looking to begin conceiving and building their projects using 3D Computer Aided Drawing.*

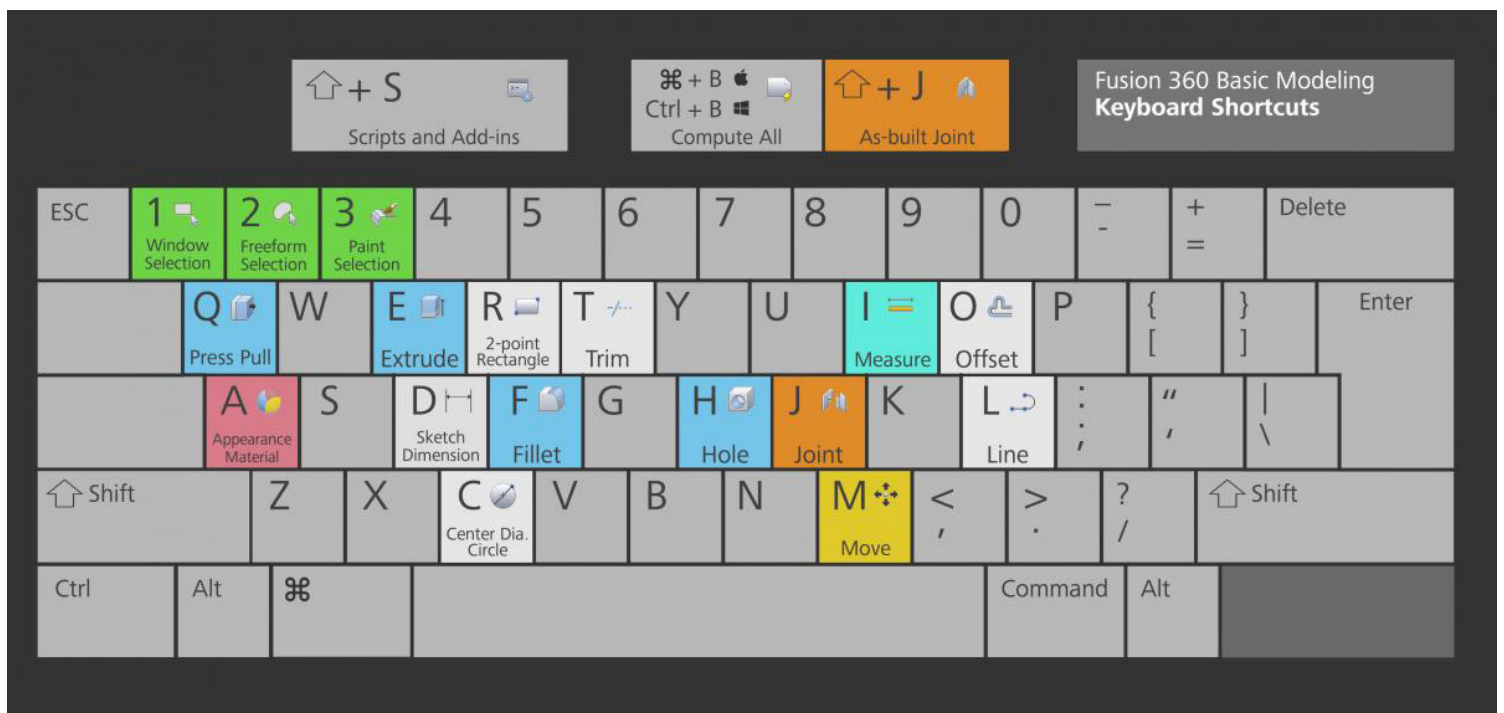


## CAPABILITIES

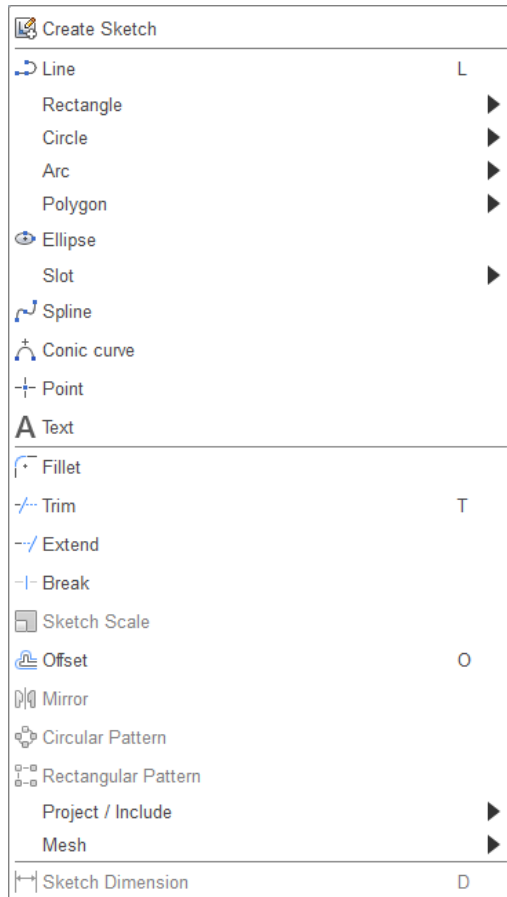
Sketch  
Model  
Sculpt  
Import  
Assemble  
Shop Drawings  
Simulate  
CAM (Computer Aided Manufacturing)  
Collaborate

## INTERFACE

Browser  
Project  
Views  
Units  
Bodies  
Sketches  
  
History Bar  
Rollback  
Suppress  
  
Toolbar and Workspaces  
Model  
Patch  
Render  
Animation  
Simulation  
CAM  
(Mesh)



# SKETCHING



Click, hold, return to point to switch between lines and arcs.

Closed profiles are labeled in orange.

Begin drawings as rough estimates.

Endlessly adjustable until dimensioned, constrained or fixed.

Define additional constraints and dimensions after sketch.

Constraints are added automatically.

Use origin when possible.

Use construction geometry for layout and relations.

Can use the same sketch for multiple features.

A fully constrained sketch's color shifts from blue to black.

# FEATURES

Break an object down into faces and features.

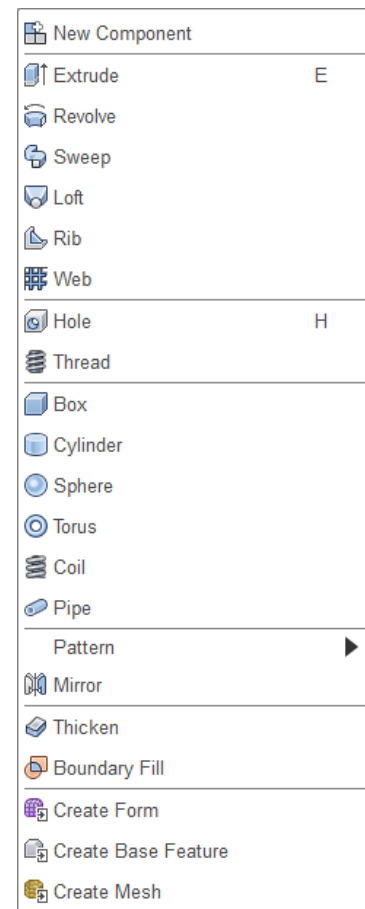
Consider Front, Side, and Top views when planning.

Reference other features and anticipate future changes.

Explore right-click menus and search commands using S-Toolbox.

Add chamfers and fillets last.

Give the instructions  
that truly tell the story  
of what you are trying  
to build.



# BOTTOM UP VS. TOP DOWN

## Bottom Up

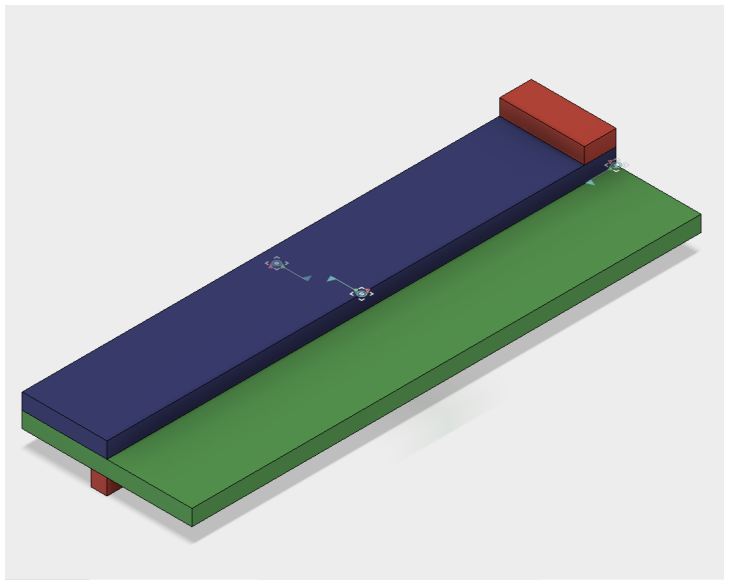
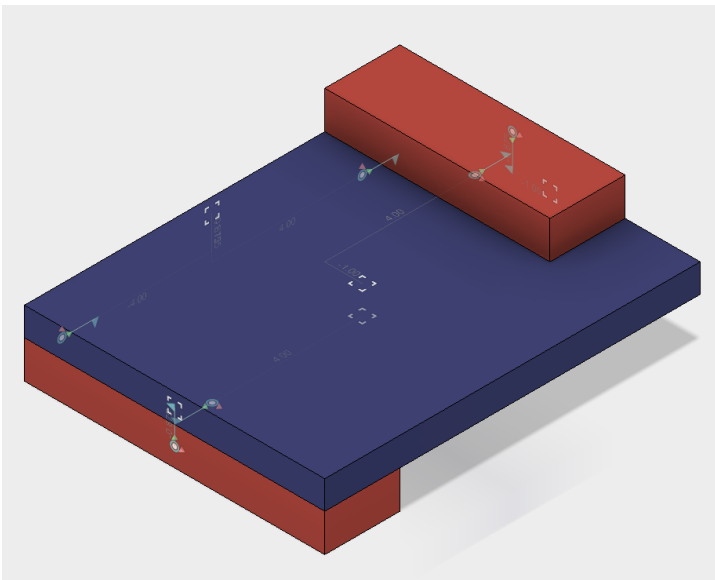
Each part is created individually, and then all the parts are inserted into an Assembly and constrained to each other.

- Dimension sketch
- Add features
- Convert bodies to components
- Use joints constraints to limit degrees of freedom
- Test assembly

## Top Down

All parts are built within the context of the assembly.

- Create a new body 'In Place', using existing features
- Add parameters and dimensions
- Create new bodies when making features
- Project geometry across parts when needed
- Use as built joints to constrain assembly
- Test assembly
- Change dimensions and watch fusion update the components.



# IMPORTING GEOMETRY

## Downloading DXF

- Insert DXF
- Extrude Sketch as Bodies
- Create Components from bodies
- Reassemble

## Download IGES

- New Design From File
- Reassemble
- Use joints between faces and move commands
- Select the component you wish to move first
- Ground base components



# PROTRACTOR

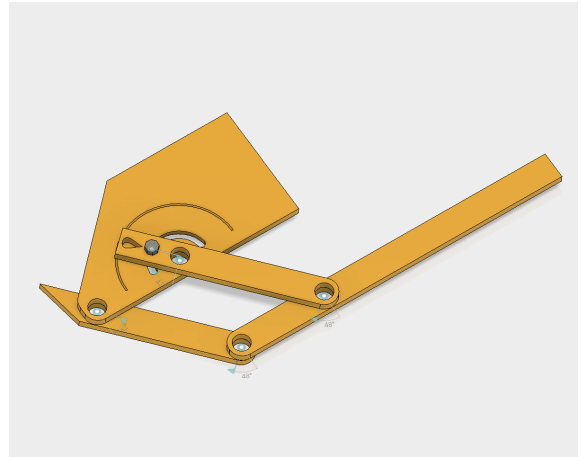
Additional Constraints

Extrudes

Fillets

Component vs Body

Planar and Revolute Joints



# CONDUIT BOX

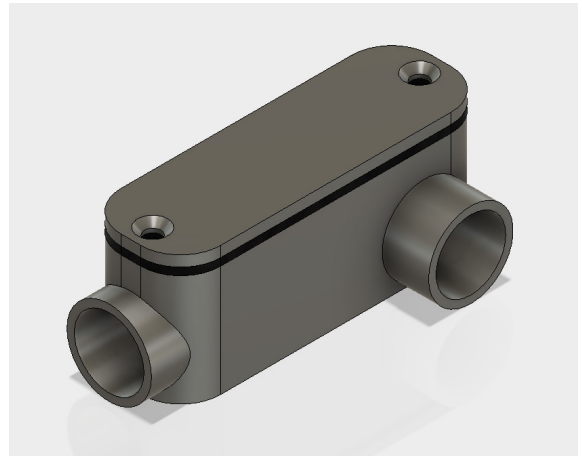
Relating sketches and features to base geometry.

Projecting and offsetting from existing features.

End conditions in feature parameters.

Mirroring

Making changes after the first iteration.



# BARB FITTING

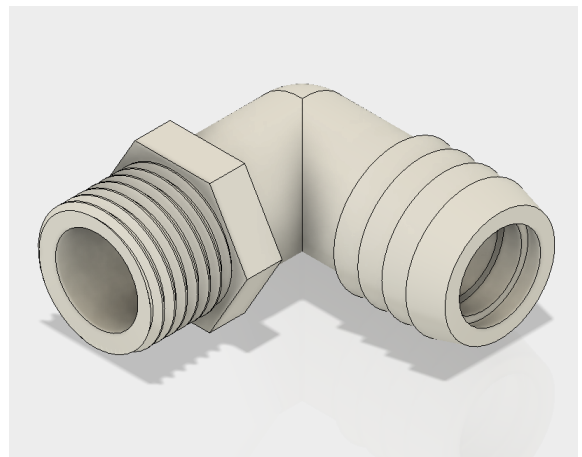
Sweep

Revolve

Threads

Shell

Parameter Changes Populate Features

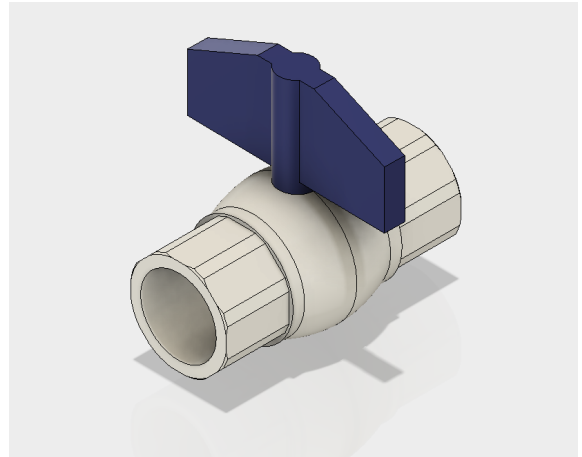


# VALVE

Top Down

Lofting

Reference Geometry

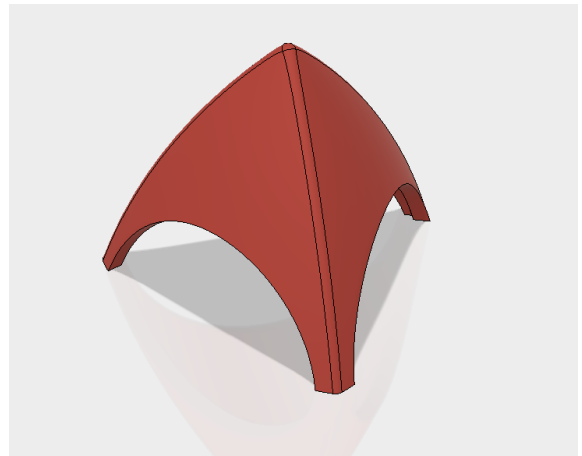


# (PAINTERS) PYRAMID

Reference Geometry

Lofting

Patterning

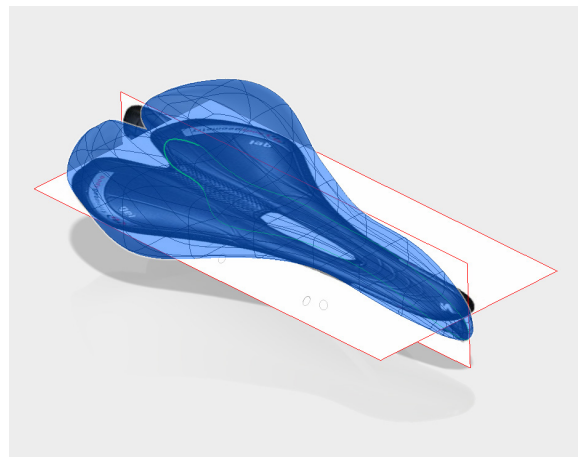


# BIKE SEAT

Working from reference images.

Sculpting with T-Splines

Project visualization - opacity control.



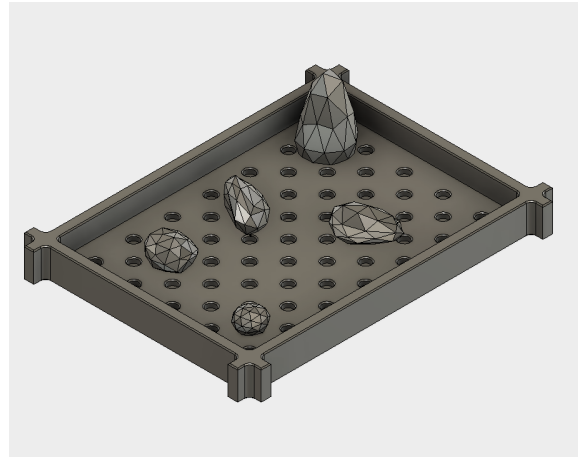
# ZEN GARDEN

Dealing with meshes

Importing

Rebuilding

Converting and Capture History

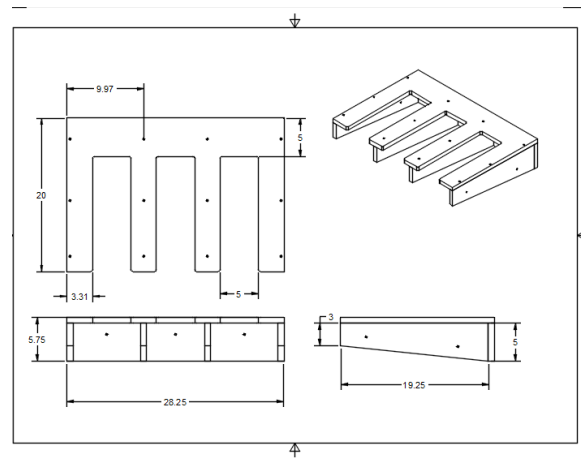


# LIGHTSTAND RACK (SHOP DRAWINGS)

From an assembly or component...

Communicate dimensions and tolerances

Relate critical information to geometry



## CONTACT

ALEX.GOSS@TXRXLABS.ORG