Goal:
Additional methods of moving from curve to surface to solid
Understanding basic geometry transformation using Rhino

Some Useful Commands
Names of commands are supposed to be descriptive.
Quick Demos on:
• Extrusion
• Trim and Split (we already covered this briefly)
• Revolve
• Sweep / Pipe
• Project (also covered before)

A Few Facts on Geometry Systems
• Cartesian geometry uses a coordinate system (x, y, z plane) to describe forms and their locations.
• Euclidean geometry classifies some forms as its basic elements: e.g. Platonic Solids which can be found under Solid menu in Rhino. (5 min tryout on different shapes: cone, cylinder, tube etc)
• Both are algebraic. That means both coordinates and shapes can be added or subtracted.

Boolean Operation
Direct algebraic operation to add and subtract objects.
(Try-out the 4 Boolean options: union, difference, intersection, split)

Quick Geometry Transformation
Rhino uses Lines/Curves as Paths for many more operations: e.g. when copy multiple objects (Array)
Demo using examples below

Real Examples
Chair example using Ellipses, Trim, Revolve, Pipe (with 2 different diameters)
A freeform staircase (railing and steps) using Pipe/Sweep, ArrayCrv, Orient…