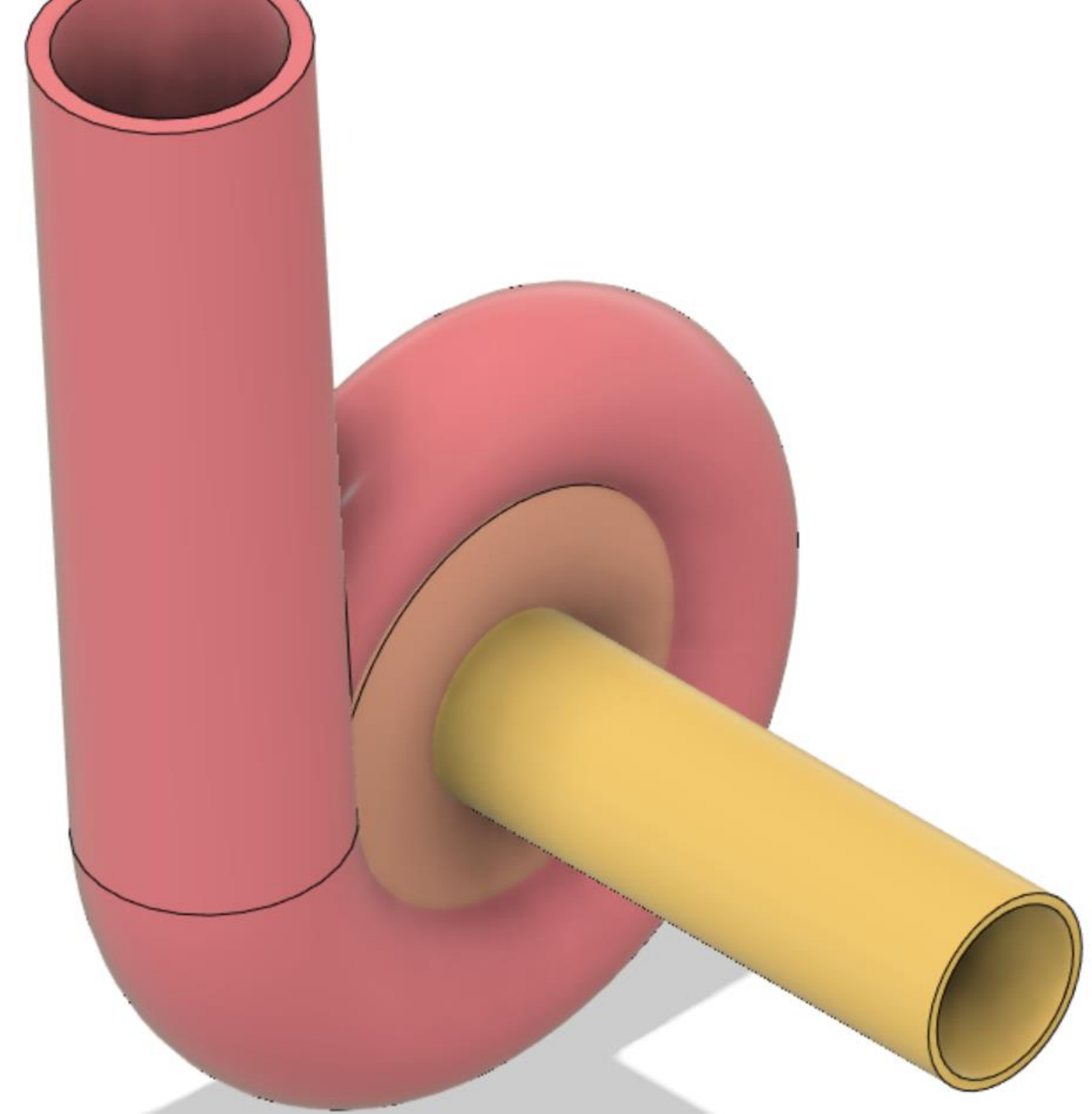


Turbines & Pumps

OA Fakinlede
oafak@s2pafrica.org; oafak@unilag.edu.ng
https://lms.s2pafrica.com/courses/continuum-mechanics-ii/

Sub-Systems Thinking

- When you live in a nation that makes practically no technological products, it is important that you think in subsystems.
- Illiterate, loud talking pundits talk about the “Nigerian Car” and such things. Can you make any of the following:
  - Car Door Handle
  - Single Cylinder Engine
  - Brake System
  - Engine Cooling System
  - Transmission System

Resources for Subsystems

- In the three-year span of our effort, those who listen, will be able to make subsystems.
  - Tensor Analysis prepares you to understand how engineers translate natural laws governing the objects you will use into mathematical equations.
  - Kinematics helps you apprehend the geometry of deformation and motion
  - Balance laws are mathematical representations of natural laws that apply to everything
  - Constitutive models are specific responses of materials to these laws.
  - Resulting equations are nonlinear partial differential equations

Software Resources

- The primary contribution of software is the numerical solution of these equations for the subsystems of interest.
  - In practical cases, your classroom solutions are only useful as starting points for understanding the real difficult equations.
  - These solutions are in the simulation packages that accompany modern CAD systems
  - In this course, we cover tensors at 300 level; Balance laws at 400 level; Constitutive laws are what you are doing in courses such as Elasticity, Plasticity, Heat Transfer, Fluid Dynamics, etc.
  - We can use simulations not only to teach; but to make, optimize and prototype subsystems incorporating these things

Virtualized Laboratories

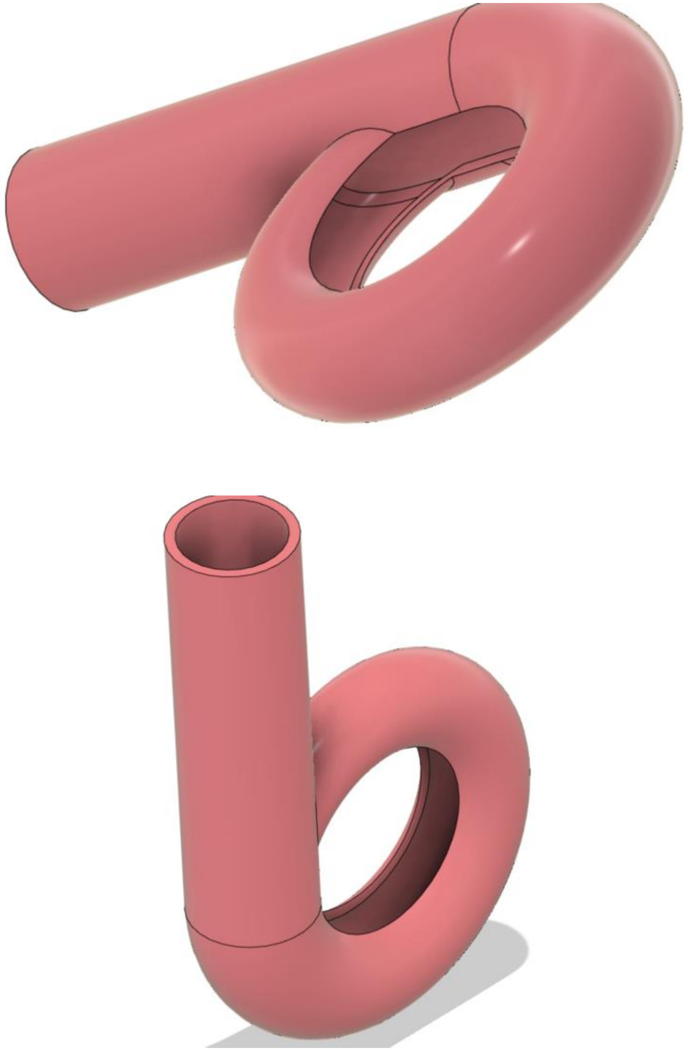
- Take our simulations classes as virtual laboratories.
  - You have access to the same software the manufacturers of the things that excite you have.
  - 50-90% of all the manufacturing intellectual labour is done by simulation. You can consider hundreds of scenarios before building the first prototype
  - That we have no labs is no longer an excuse to be unproductive. Use what we have to the limit; then we can complain afterwards.

The Turbine & Centrifugal Pump

- The centrifugal pump is (almost) everywhere in Nigeria.
  - How many do we buy yearly!
  - Are they made by angels or human beings?
  - Here are our sisters!
  - You can watch this video to learn details of how centrifugal pumps work
- Turbines: Same components in reverse.
  - If you can make a centrifugal pump, you can make a turbine.

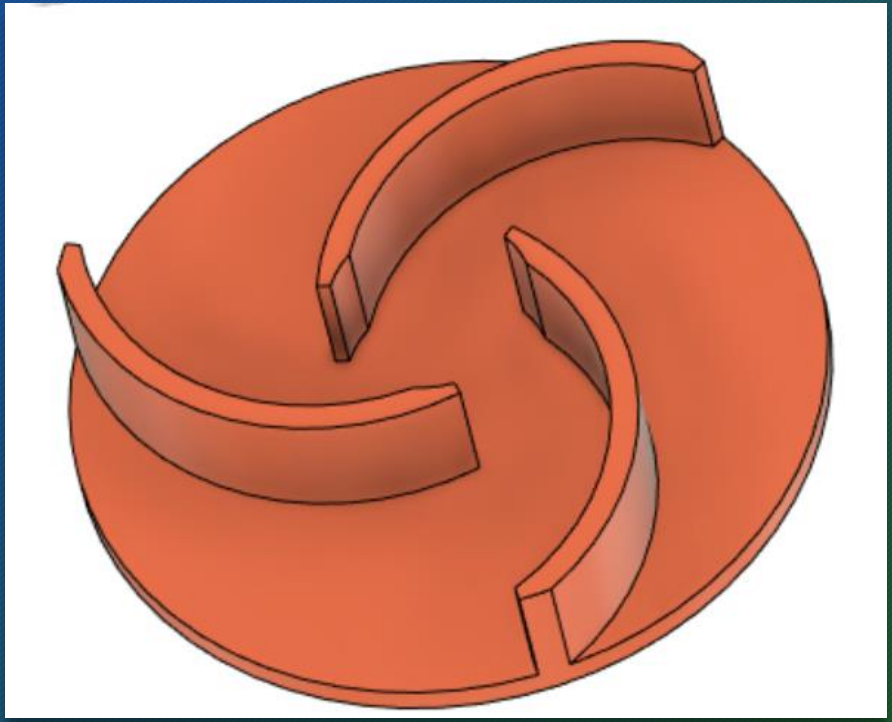
The Volute

- Here are two views of a typical "Volute"
  - Essentially a hollow metallic vessel with a decreasing spiral shape that ends in a straight protrusion as shown.
  - It is open at the protruding end, the inner part of the spiral vessel is opened to receive a cylindrically shaped rotating part chiefly made up of the Impeller.
  - The volute is the passive or static part, the impeller is where the actions take place.
- There are many other possible ways to shape a volute. Google this and see the tens or hundreds of volutes that have been designed.



The Impeller

- Here are perhaps some of the simplest ways an impeller may be shaped.
- The blades can come in a variety of ways, numbers and variableness.
- The disk is usually attached to be moved by an engine (pump) or to drive a shaft (turbine)
- It is obvious that the shaping and orientation of the blades will affect the way the pump behaves.



Your Virtual Laboratory

We can study the turbine or compressor by doing the following:

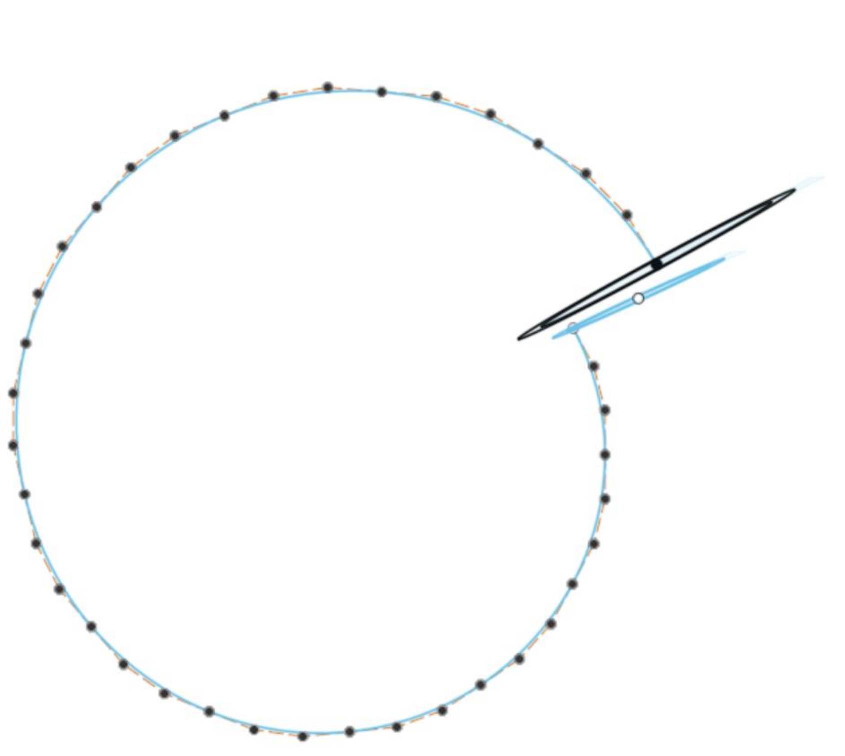
- Change the number and shapes of the blades
- Rearrange the curvature and narrowing of the volute
- Adjust the size of the disk

These are things that may even be difficult in the best laboratories to achieve.

- With Computational Fluid Dynamics Simulation, we can do all these virtually just the same way any modern manufacturer of a turbine would!

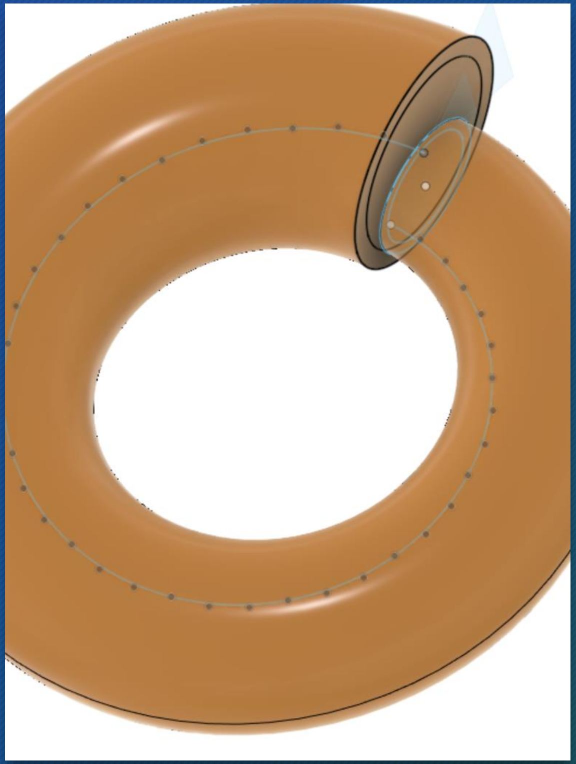
First Order of Business

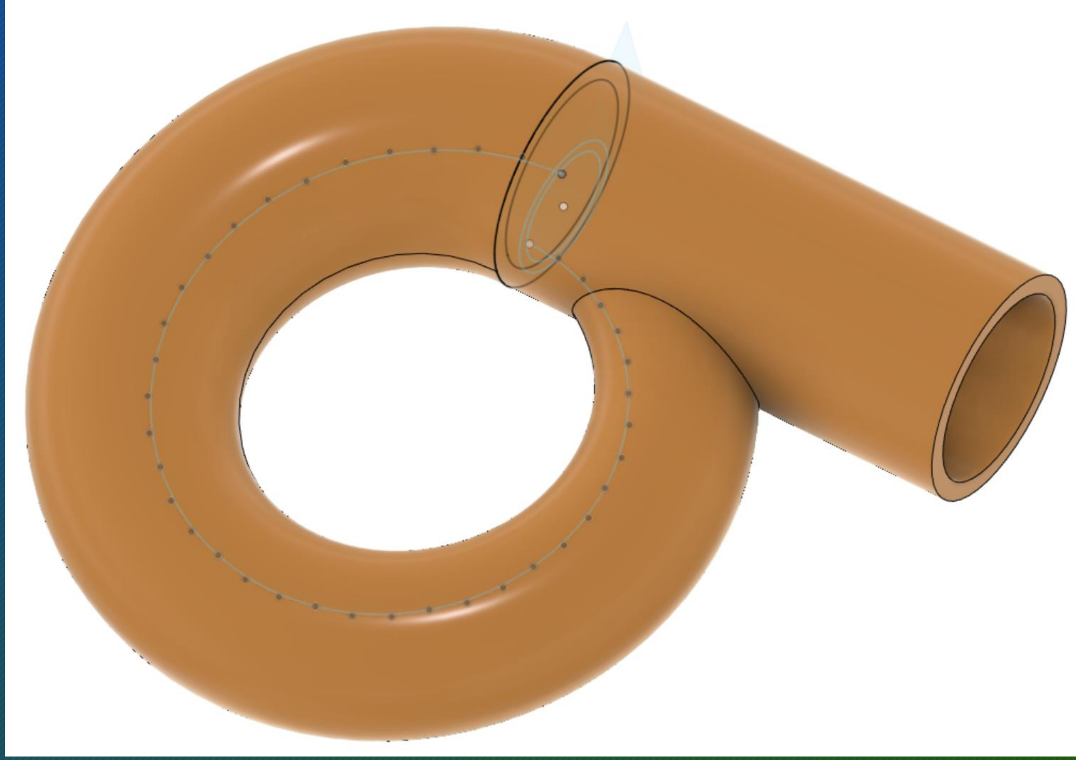
- Draw the Volute.
  - This turned out to be more difficult that I had expected:
  - Problem one is that Fusion 360, unlike Autodesk Inventor, for example, does not directly support computed splines.
- You have two options:
  1. Use the “coil” command to make a spiral using a prismatic cross section, you can take the spirals generated.
  2. Write a code using Python or C++ into the Fusion 360 API.
- Either gives a spiral to loft two unequal annuli as shown to get



First Order of Business

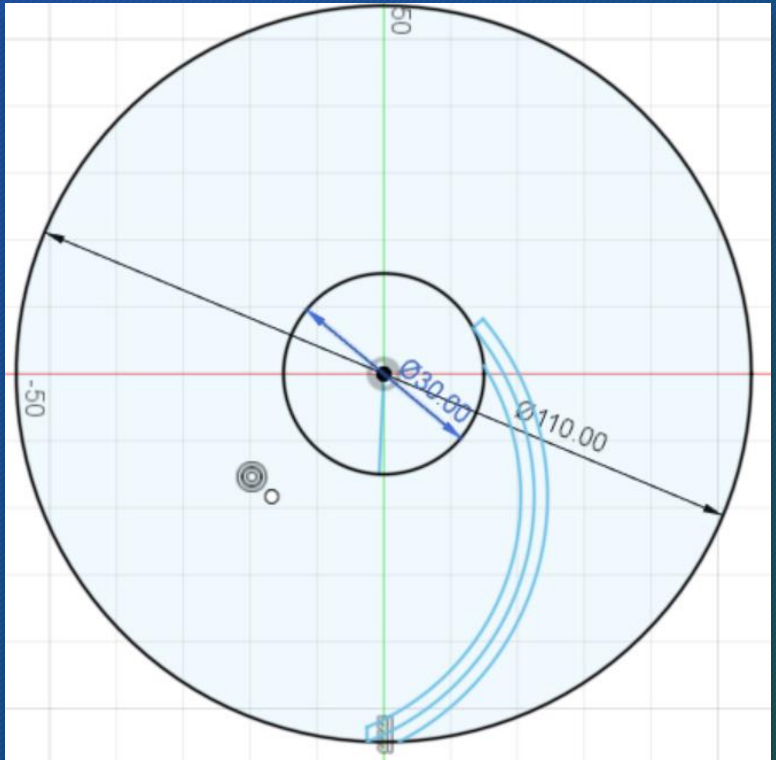
- Either gives a spiral to loft two unequal annuli as shown to get the figure here.
- Note that you will need to loft a second time to create the hollow.
- Several lofts and intersections later, you reach the complete volute.
- Straight portion is further elongated for developed flow:
    - More about that in your fluid mechanics classes

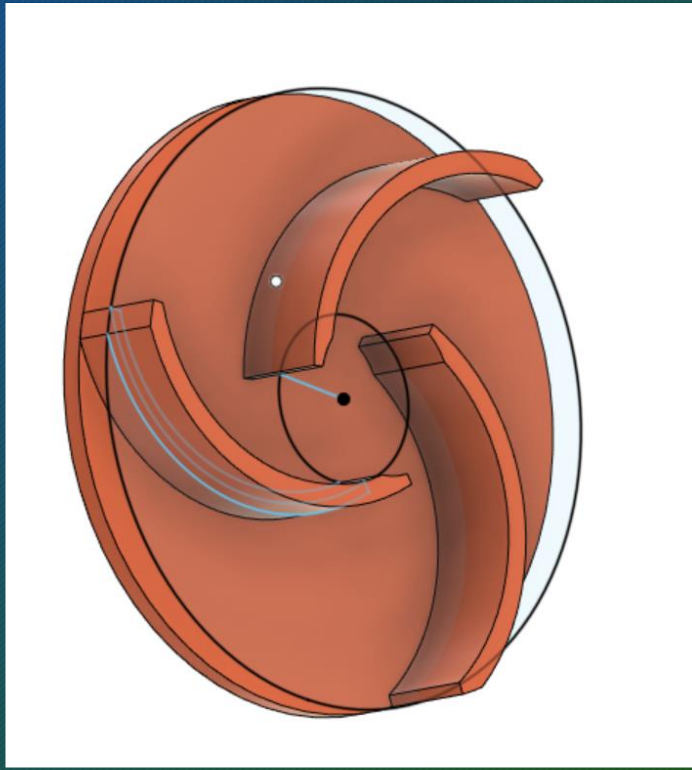




Impeller

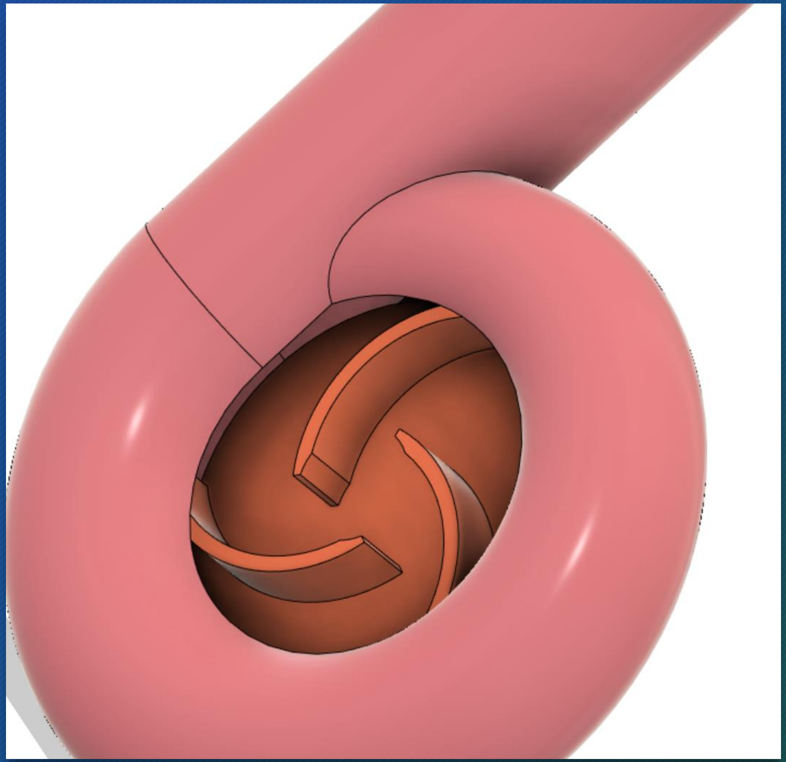
- Making the impeller is a simple circle extrusion with the extrusion of a 3-point doubly offset arc shown here.
- We can then decide how many blades we want and create a circular pattern of them.

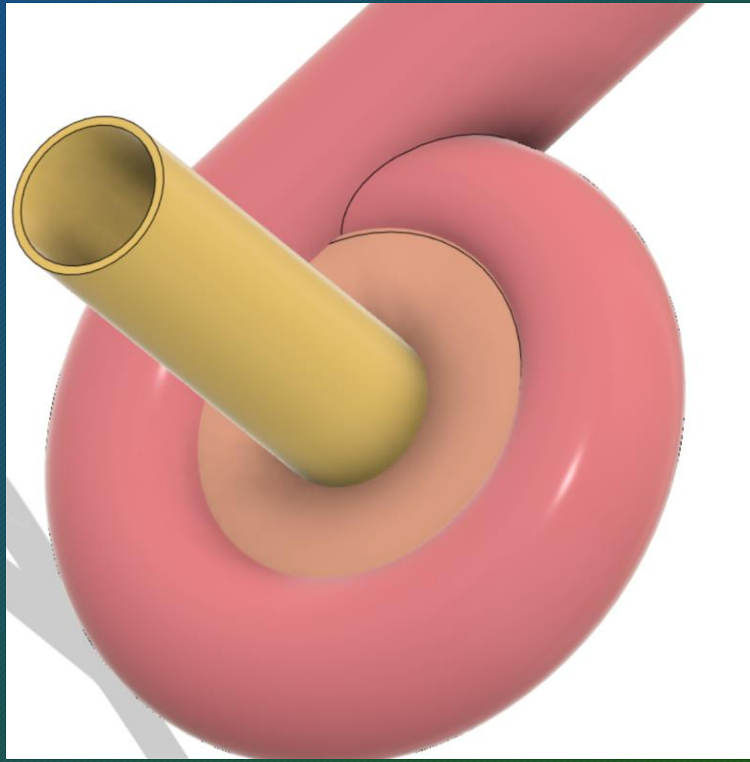




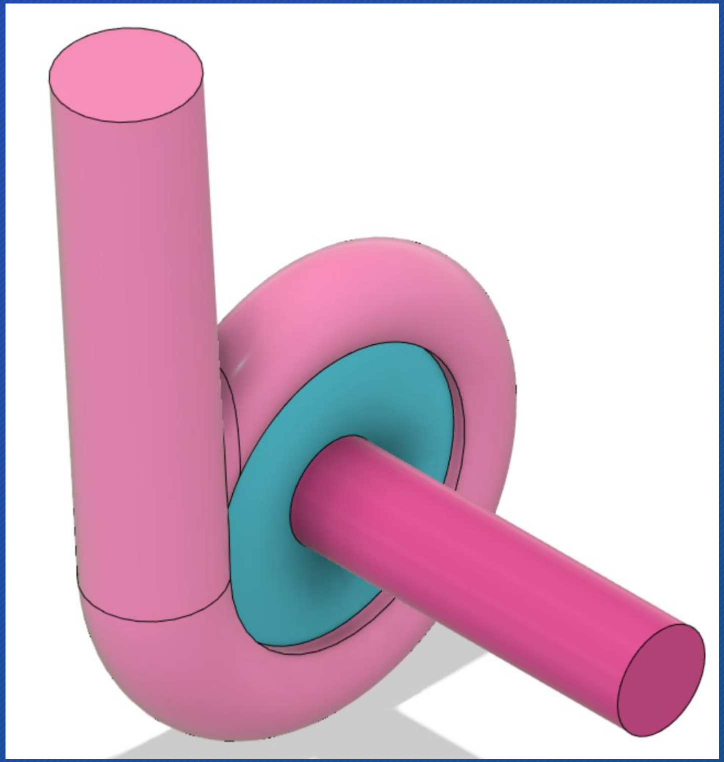
The Assembly

- We make covers to the front and back and proceed to assembly using a rotational joint between the impeller and the volute body
- Notice the elongations for the water intake and water outflow.
  - Demonstrate the joints
  - We need fully developed flow that you will only appreciate after the theory classes.

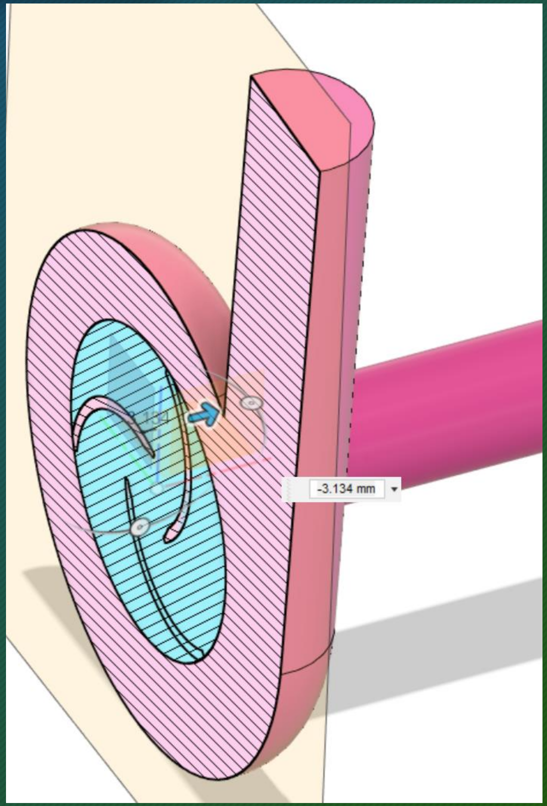




Work Stages

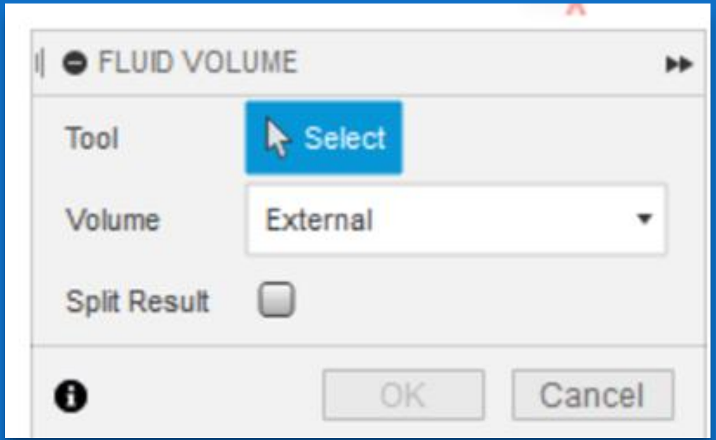


STAGE	WORK DONE
I	Create the Physical Model: Just completed
II	Prepare the Fluid Body: Create the moving Fluid and the Rotating Region
III	Enter the Simulation Data: Initial, Boundary conditions & Material Properties
IV	Solve: Finite Element Solution of the resulting Differential Equations
V	Post Processing: Interpreting the Results



Creating Fluid Volumes

- Selecting Simplify takes you to a different environment
  - In this environment, you can turn your model to different regions of “Fluid volumes”.
  - A fluid volume is a region of engulfed fluid. For example to make an inlet fluid volume that you see in purple, we first create a water-tight region as tools and Fusion 360 will turn that to a fluid volume.
  - A fluid volume may be internal or external



Rotating Region

- The rotating region is the combination of the impeller and the fluid region that moves with it.
  - In reality, the fluid moves away from the impeller by centrifugal force action. In the model, it appears as if they constitute a solid rotating region. This is how the software works to identify what to apply the fluid motion equations to

Simulations Data

- After creating the fluid regions, you can launch, from inside Fusion 360, the Autodesk CFD simulation software.
  - You specify your units; material properties, rotating region properties, etc.
  - You also specify the axis of rotation, speed of rotation – if known (it may not be known)
  - You can specify known inlet conditions: Here we know the inlet pressure but not the velocity. The latter will be computed.
  - You also specify the mesh sizes and regional diversity of meshes.

Solving

- What happens in this step is the solution of the partial differential equations of the flow:
  - These are the well-known Navier-Stokes Equations for real linear fluids.
  - Autocad CFD can simulate non-linear fluids with much more complicated equations than the Navier Stokes.
  - You can simulate a blood pumping machine designed for a blood bank for example!
  - You may want to design a pump for pumping hot coal tar in a bitumen factory.