

## INTRODUCTORY OCEAN ENGINEERING

**CE 358 (KINNAS) - FALL 2025**

### TERM PROJECT - PART A<sup>1</sup>

- Must be done and submitted separately by each student**
- Counts for 6% of the total grade**

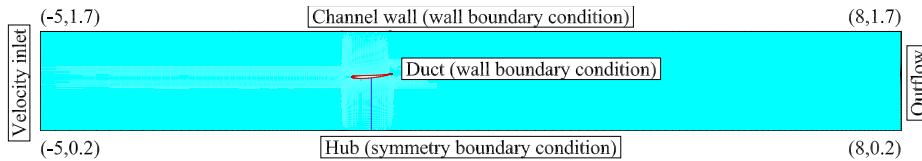


Fig1(a). An axisymmetric duct is placed inside a cylindrical channel.

(Coordinates of the 4 corners of the computational domain are given. All values are non-dimensional by the radius of the turbine blade)

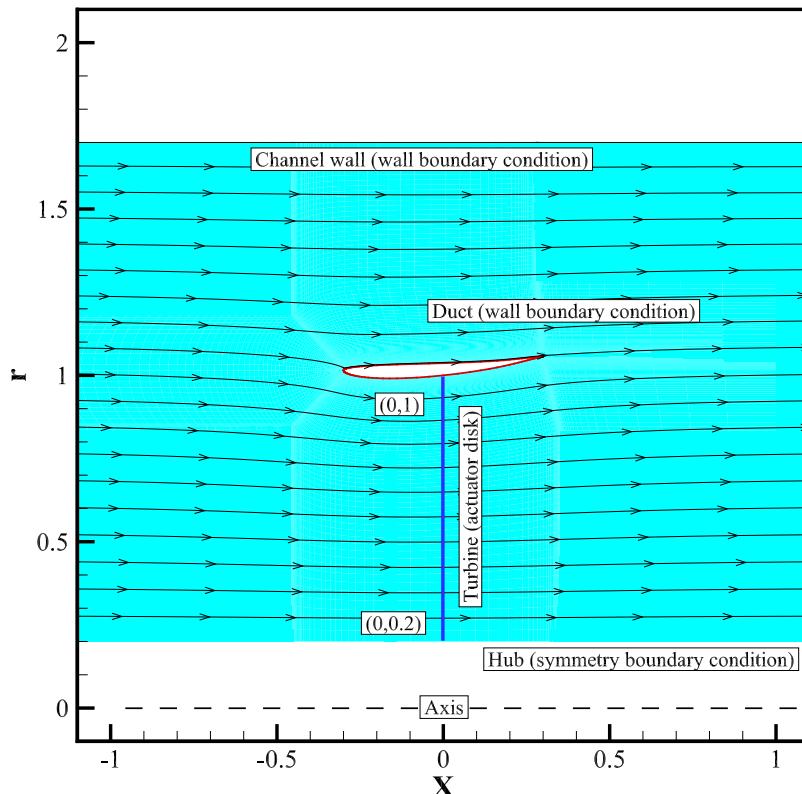


Fig1(b). The streamlines around the duct.  
(Blue line: actuator disk, which models the turbine with a pressure jump<sup>2</sup>).

The power output of a tidal turbine is proportional to the cube of the inflow velocity. For example, if the inflow is increased by 20%, the produced power will increase by 72.8%. In this project, you are going to use an axisymmetric duct to accelerate the inflow at the turbine plane (shown as a purple line in the figure above), and thus increase the produced power. In Part A, you will use the commercial computational fluid dynamics (CFD) software *ANSYS/Fluent* to study the effect of the duct itself without

<sup>1</sup>©S.A. Kinnas, 2025

<sup>2</sup>In Part A, we will not consider the turbine.

the turbine. In Part B, you are going to design a turbine to work with a given duct geometry, build it by the 3D printers, and test it in CAEE's water flume.

As shown in the figures above, a duct is placed in freshwater ( $\rho = 999.965 \text{ Kg/m}^3$ ), and a uniform inflow is coming from the left towards the duct with a constant speed  $U_{in}$ . The channel is assumed to be cylindrical<sup>3</sup> which simplifies the problem to be 2D axisymmetric, i.e., the cross section shown in Fig.1(b) will rotate by  $360^\circ$  around the axis. The chord length of the duct  $c_{duct}$  is equal to 60% of the radius of the turbine blade. The coordinates of the computational domain are given in the figure above. A package which contains the input files *fluent.msh* and journal files for running *ANSYS/Fluent* can be downloaded from the term project web site.

On the term project website you may also find a **tutorial with step-by-step instructions** on how to run *Fluent* (PDF file). The sample run in the ANSYS/Fluent tutorial, is for the above duct shape at  $U_{in} = 1.0 \text{ m/s}$ , with  $\rho = 999.965 \text{ Kg/m}^3$  and  $\mu = 0.001519 \text{ Pa sec}$ . **VERY IMPORTANT: You should read the tutorial very carefully, and you must recover on your own the answers for the sample run, BEFORE you do Part A of the term project.**

Consider the average temperature of the water to be  $5^\circ$  Celsius. For the given duct you are asked to do the following, using the same given grid for all cases. Instructions on how to do each task are provided in the tutorial. **ONLY computer generated and fully annotated plots will be accepted:**

- a) Determine and plot  $C_{D,friction}$ ,  $C_{D,pressure}$ , and  $C_{D,total}$  acting on the duct as a function of the velocity<sup>4</sup>. Do this for a range of velocities between 0.1-1.5 m/s with at least 13 points in between (i.e. at least 15 points, including the ends). **(40 points)**
- b) Determine and plot the total drag force acting on the duct as a function of velocity. Use the same velocities as in the previous question. **(20 points)**

**Do the next 3 questions for a given velocity (=last 3 digits/letters of your UT EID number)  $\times 0.001 + 0.5$ , in m/sec; - 0:VW; 1:X; 2:YZA; 3:BC; 4:DEF; 5:GHI; 6:JKL; 7:MNO; 8:PQR; 9:STU - e.g. if your UT EID is XXXX3sa the velocity should be  $392 \times 0.001 + 0.5 = 0.892 \text{ m/sec}$ )**

- c) Plot the pressure coefficient along the duct ( $C_p$  vs.  $x$ ) **(15 points)**. Find the pressure coefficient at the stagnation point and explain its value (*Hint: Bernoulli equation*) **(5 points)**.
- d) Plot the axial velocity profile along  $r$  direction ( $u$  vs.  $r$ ) at five  $x$  locations ( $x = -0.4, 0.0, 0.35, 0.6, 1.5$ ). Compare the four curves and comment on their differences or similarities. **(20 points)**
- e) Bonus Question. Find the average velocity (in  $\text{m/s}$ ) at the turbine location ( $x = 0$ ) and the flow rate ( $\text{m}^3/\text{s}$ ) through the actuator disk by numerical integration of the resulting velocity profile by Fluent (*velocity.dat*). Note that the integral has to be carried over a hollow circular disk (from  $r = 0.2$  to  $r = 1$ ). Use trapezoidal rule, or any other integration scheme. By how much you expect the produced power of the turbine to increase due to the presence of the duct? **(10 points)**

**IMPORTANT NOTES: For full credit please provide details of your procedure and the parameters you used when running Fluent, rather than just the plots. Each student is expected to submit his/her own report and graphs in Part A. Identical reports and/or graphs submitted by two or more students will result into a significant grade reduction for all these students.**

---

<sup>3</sup>The flume section is 0.3 m wide, and the water level is set at 0.3 m as well; Thus, the actual section is a 0.3 m square, which is approximated with a circle of the same sectional area.

<sup>4</sup>In Fluent “viscous  $C_D$ ” is actually what we defined in class as  $C_{D,friction}$ .