Air Breathing Propulsion Field Exam Questions

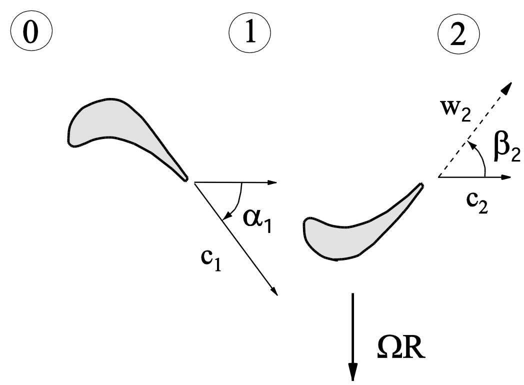
Department of Aeronautics & Astronautics
January, 2018

*Do Problem 1 and **either** Problem 2 on internal flow **or** on compressible flow*

Problem 1 (Weighted 50%)

You are asked to assess a turbine design for mechanical integrity to avoid a potential uncontained engine failure. The specific question is whether the turbine runaway speed is below the disk burst speed should the shaft connecting the turbine to the compressor shear off and break.

The following assumptions can be made. The flow through the nozzle guide vanes (NGVs) is choked at the vane passage exit (c_1 is sonic). The flow angles α_1 and β_2 are 60 and 45 degrees respectively and the mean radius R is 0.3 m. The turbine inlet stagnation temperature is 1500K. The axial velocity through the turbine stage is constant. At design speed the flow leaves the stage without swirl. The three stations, 0, 1, 2, are upstream of the NGVs, between the rotor and stator, and downstream of the rotor.



a) Find the NGV exit velocity c_1 and relative rotor exit velocity, w_2, at design conditions.

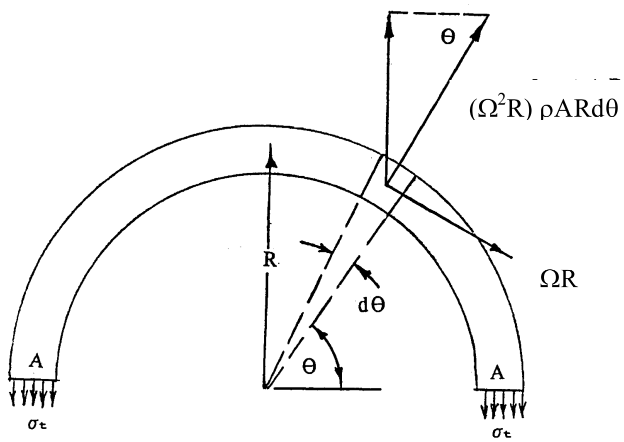
b) For the given flow angles at design conditions, what is the rotor shaft speed in rpm?

Suppose that the shaft connecting the turbine and the compressor suddenly breaks, with all the turbine inflow conditions remaining unchanged (stator vanes are still choked). The turbine speeds up and is freely rotating (bearing friction can be neglected). Assume for now that structural integrity of the disk and blades is maintained independent of rotor speed.

c) Will the turbine shaft rotate infinitely fast or will it come to a steady rotation rate? Why or why not? An explanation using first principles is expected.

d) Find the speed at which the turbine will end up rotating.

Next, consider the turbine disk to be an assemblage of concentric rings one of which is shown below. Neglecting the pull of the blades and assuming uniform stress, the burst speed is reached when the hoop stress, σ_t , reaches the material's ultimate strength.

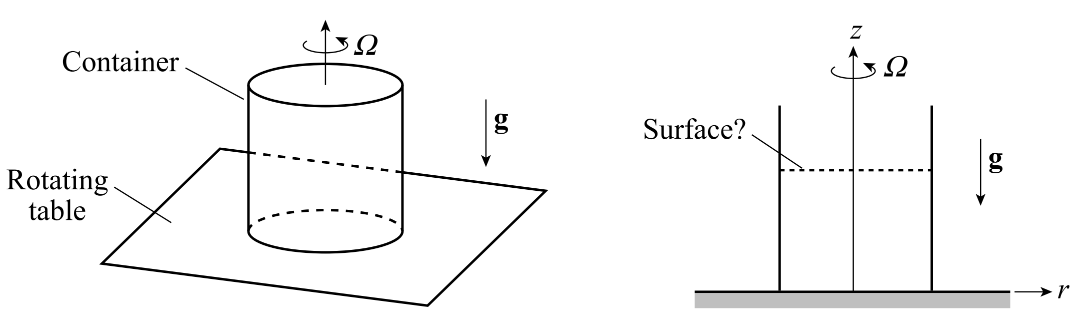


e) For the concentric ring of radius R above, find the hoop stress, σ_t, as a function of rotor speed.

f) Returning to the key issue posed above, will the disk burst if the shaft shears? You can assume R = 0.2 m for the disk radius and σ_{ult}/ρ = 40 kPa/kg-m^3 for the ultimate specific material strength.

Problem 2 – Internal Flow Option (Weighted 50%)

A heavy liquid in a cylindrical container of radius R sits on a table that rotates with angular velocity of magnitude Ω, as below. The liquid has uniform density, ρ. The axis of the container coincides with the axis of rotation, and both are aligned with the gravitational force, which points downward. The top of the container is open to the atmosphere. The container has been rotating long enough so that all the liquid has angular velocity Ω about the axis of rotation.



Liquid in a rotating container

a) What is the radial pressure gradient in the liquid?

b) What is the vertical pressure gradient in the liquid?

c) What is the shape of the surface?

d) What is the pressure distribution on the bottom of the container?

e) What will be the behavior of small particles dropped into the container that make their way to the bottom? Do they move outward, move inward, or stay where they are dropped?

f) If there were *two layers* of liquid, one of ρ_1 and one of ρ_2, would the surface of the upper liquid have the same shape as the interface between the two liquids?

g) Is there a quantity that is constant along the surface or along the interface?

Make any approximations you think necessary, but be prepared to provide justification for the assumptions.

Problem 2 – Compressible Flow Option (Weighted 50%)

Air flows, *adiabatically with friction*, in a channel of circular cross section. The flow is *subsonic*, and the friction coefficient, $C_f = \tau_w/(\rho u^2/2)$, wall shear stress/dynamic pressure, is constant. The channel is designed so *the Mach number is constant along the duct.*

a) Does the area of the duct increase, decrease, or stay constant with distance along the duct?

b) If the area is not constant, what area variation with distance along the duct is needed to achieve the desired condition? (An explicit expression in terms of parameters such as Mach number, coefficient of friction, etc., is wanted.)

c) What is the stagnation pressure variation with distance along the duct?

d) Is there a net axial force on the duct surface? If so, in which direction does it point?

e) If the flow were supersonic at constant Mach number, would the change in area with distance have the *same sign* as in subsonic flow?

Notes:
1) It is not necessary, but it may be helpful, to think about this problem in terms of the behavior of the corrected flow per unit area along the duct.

2) If we define dA_w as the "wetted area" over which the skin friction acts, for a small length, dx, it may be helpful in the algebra to write the skin friction term for the circular duct as

$$\tau_w dA_w = C_f(\rho u^2/2)\frac{4Adx}{d_H},$$

where d_H is the "hydraulic diameter" defined as $4A$/wetted perimeter.