Table of Contents

0

```
close all;
clear all;
clc

addpath("C:\joshFunctionsMatlab\")
```

toolbox

```
clear all;
DM = importdata("DMout1.mat");
R = 287;% J/kg.K

n = 100;
M = linspace(0,1,n);

Tr = zeros(1,n);
Pr = zeros(1,n);
rhor = zeros(1,n);

for i = 1:n
    [Tr(i),Pr(i),rhor(i)] = joshIsentropicToolbox(M(i));
end

figure
hold
plot(M,Tr.^-1)
plot(M,Pr.^-1)
plot(M,rhor.^-1)
legend("Tr","Pr","rhor")

gamma =1.4 ;
a =@(T) sqrt(gamma*R*T);
Ma = @(U,T) U/a(T);
Ttotal = DM(1).T_amb;
Ptotal = DM(1).Amb_Press;
rhototal = 1.204; % kg/m^3
Cp = 1005 ;%J/kg.k

for i=1:length(DM)
    for j = 1:length(DM(i).V_fromA)
```

```
        DM(i).Ma(j) = Ma(DM(i).V_fromA(j),Ttotal);
    end
end

for i = 1:length(DM)
    for j = 1:length(DM(i).Ma)
        [DM(i).Tr(j),DM(i).Pr(j),DM(i).rhor(j)] =
 joshIsentropicToolbox(DM(i).Ma(j));
        DM(i).T_isen(j)=Ttotal/DM(i).Tr(j);
        DM(i).P_isen(j)=Ptotal/DM(i).Pr(j);
        DM(i).rho_isen(j)=rhototal/DM(i).rhor(j);
        DM(i).error(j) = (DM(i).P(j)-DM(i).P_isen(j))/DM(i).P_isen(j);
        DM(i).ds(j) = Cp*log(DM(i).Tr(j))-R*log(DM(i).Pr(j));
    end
end

disp("------------ISF------------")
disp("My work has the following results:")
disp("I calculated that as compared to the measured values of P, isentropic
 relations find the following errors as a percent and delta s in kJ/kg.K in
 that order at each station:")
disp("run1:")
disp("error:")
disp(DM(1).error*100)
disp("delta s:")
disp(DM(1).ds)

disp("run2:")
disp("error:")
disp(DM(2).error*100)
disp("delta s:")
disp(DM(2).ds)

disp("run3:")
disp("error:")
disp(DM(3).error*100)
disp("delta s:")
disp(DM(3).ds)

disp("run4:")
disp("error:")
disp(DM(4).error*100)
disp("delta s:")
disp(DM(4).ds)

disp("This loss of entropy generally less than a percent seems to be plenty
 within limits to call the windtunnel isentropic. Most likely the flow in the
 center of the tunnel is closer to isentropic flow than that on the edges,
 since we are assumming that the loss of entropy comes mostly from friction
 with the walls and possibly some leaking in the wind tunnel which will cause
 a loss of total pressure resulting in a measured loss of entropy.")

disp("I calculated the following density ratios based on Isentropic
 assumptions:")
```

```
disp("run1:")
disp("rhoTotal/rho:")
disp(DM(1).rhor)

disp("run2:")
disp("rhoTotal/rho:")
disp(DM(2).rhor)

disp("run3:")
disp("rhoTotal/rho:")
disp(DM(3).rhor)

disp("run4:")
disp("rhoTotal/rho:")
disp(DM(4).rhor)

disp("These values seem to be close to the plots of rho ratio vs Ma, the
 highest Ma is around .1 and by the graph this should yeild a rho ratio of
 about 1/.99 or 1.005. With such low mach numbers it is slightly hard to tell
 however, rho almost doesn't change and incompressible is probably not a bad
 assumption for this process in general.")
```

```
Current plot held
------------ISF------------
My work has the following results:
I calculated that as compared to the measured values of P, isentropic
 relations find the following errors as a percent and delta s in kJ/kg.K in
 that order at each station:
run1:
error:
   -0.0093    0.0420    0.0038   -0.0277   -0.0262

delta s:
   1.0e-04 *

    0.0082    0.7539    0.7539    0.3805    0.1625

run2:
error:
   -0.0255    0.1476   -0.0223   -0.1366   -0.1079

delta s:
   1.0e-03 *

    0.0028    0.2546    0.2546    0.1285    0.0549

run3:
error:
   -0.0466    0.2737   -0.1253   -0.3505   -0.2561

delta s:
   1.0e-03 *

    0.0051    0.4711    0.4711    0.2378    0.1016
```

run4:
error:
```
   -0.0680    0.7828    0.0611   -0.4741   -0.3879
```

delta s:
```
    0.0000    0.0012    0.0012    0.0006    0.0003
```

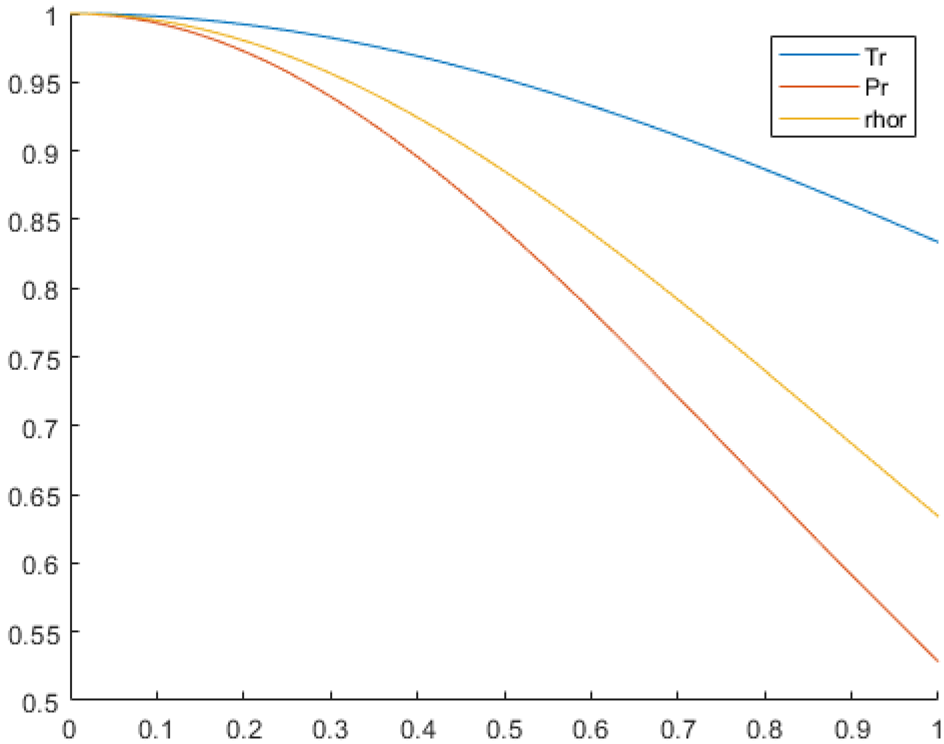
This loss of entropy generally less than a percent seems to be plenty within limits to call the windtunnel isentropic. Most likely the flow in the center of the tunnel is closer to isentropic flow than that on the edges, since we are assumming that the loss of entropy comes mostly from friction with the walls and possibly some leaking in the wind tunnel which will cause a loss of total pressure resulting in a measured loss of entropy.
I calculated the following density ratios based on Isentropic assumptions:
run1:
rhoTotal/rho:
```
    1.0000    1.0004    1.0004    1.0002    1.0001
```

run2:
rhoTotal/rho:
```
    1.0000    1.0013    1.0013    1.0006    1.0003
```

run3:
rhoTotal/rho:
```
    1.0000    1.0024    1.0024    1.0012    1.0005
```

run4:
rhoTotal/rho:
```
    1.0001    1.0062    1.0062    1.0031    1.0013
```

These values seem to be close to the plots of rho ratio vs Ma, the highest Ma is around .1 and by the graph this should yeild a rho ratio of about 1/.99 or 1.005. With such low mach numbers it is slightly hard to tell however, rho almost doesn't change and incompressible is probably not a bad assumption for this process in general.



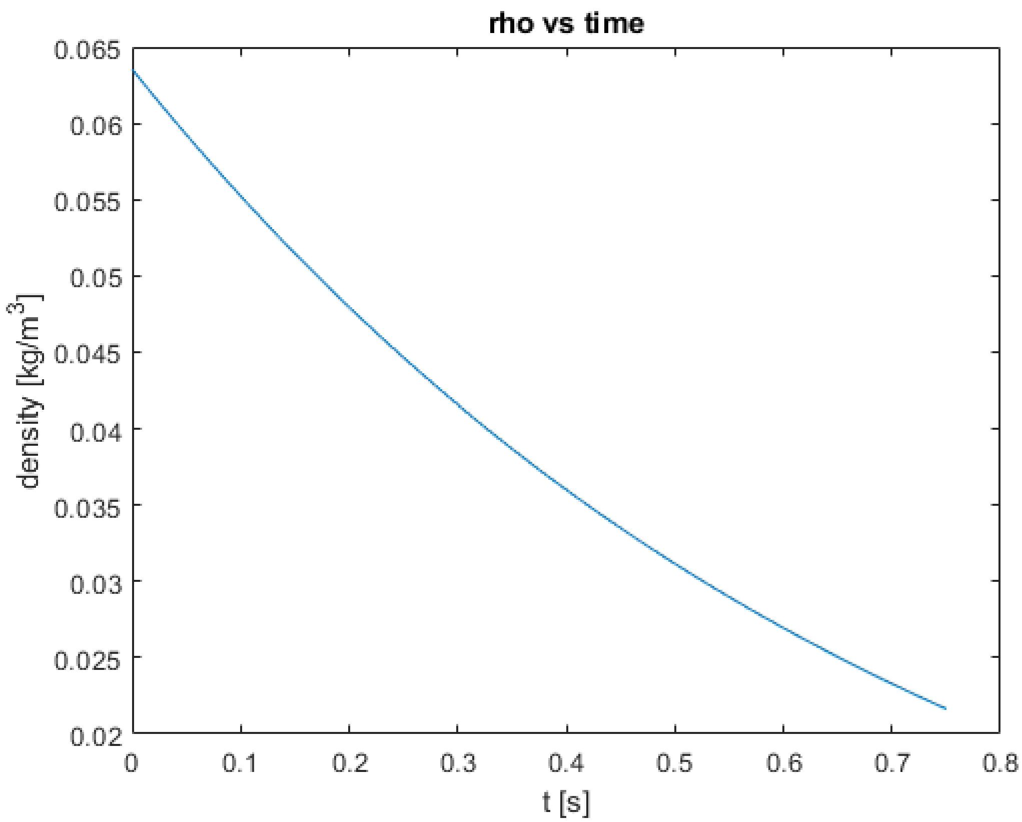
FK

FK was completed by hand, see PDF

CM1

```
clear all
Ue = 15;%m/s
L = 10;%m
Xdot = @(t,X) X*Ue/L;

dbulb = .15;
dcond = .1;
V0 = (4/3)*pi*(dbulb/2)^3;
P = 199325;
rho = .9108;
m = rho*V0;

tspan = [0 .75];

options = odeset('RelTol', 1e-8,'AbsTol',1e-8);
X0 = 0;
[t1,X1] = ode45(Xdot,tspan,X0,options);
X0 = 3;
[t2,X2] = ode45(Xdot,tspan,X0,options);
```

```
rho = @(v) m/(V0+v);
v = @(x) pi*(dcond/2)^2*x;

for i = 1:length(X2)
    rhov(i) = rho(v(X2(i)));
end

figure
plot(t2,rhov)
title("rho vs time")
xlabel("t [s]")
ylabel("density [kg/m^3]")
```



CM2

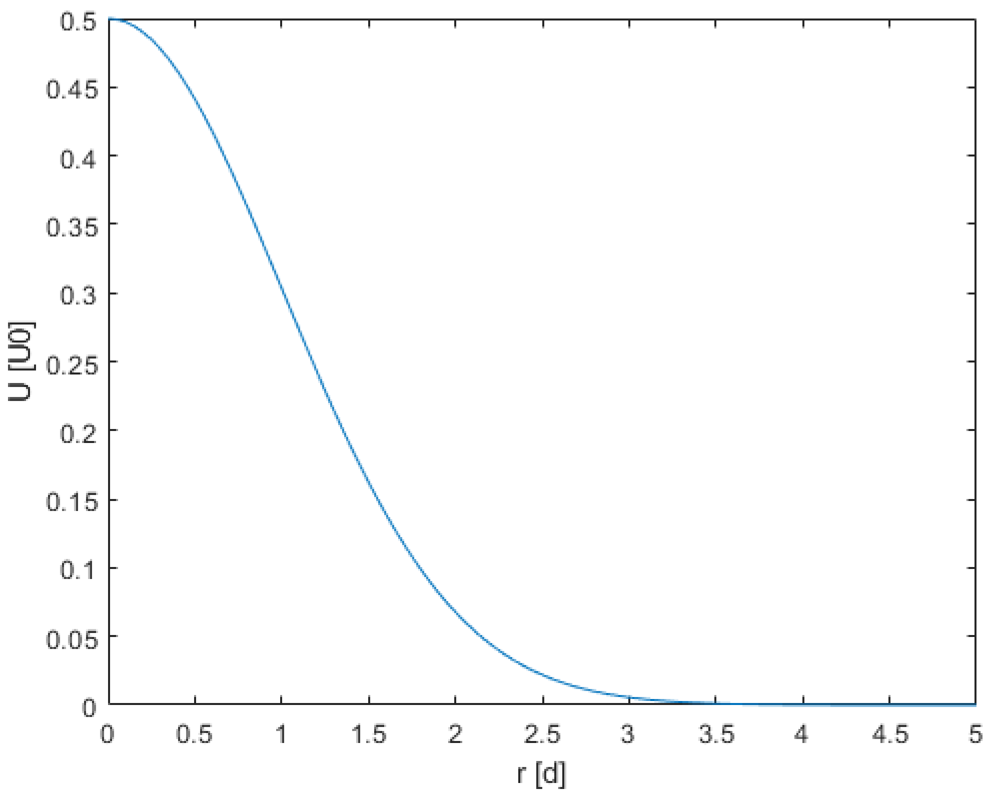
```
clear all

syms U0 d rho% U0 is [m/s] and d is [m] but i will consider they themselves to
 be units of length and velocity respectively
A0 = pi*(d/2)^2; % [d^2]
vdot0 = U0*A0; % mass flow rate at exit in terms of U0 and d
mdot0 = vdot0*rho;

% in this case below, x means times or function handle name
```

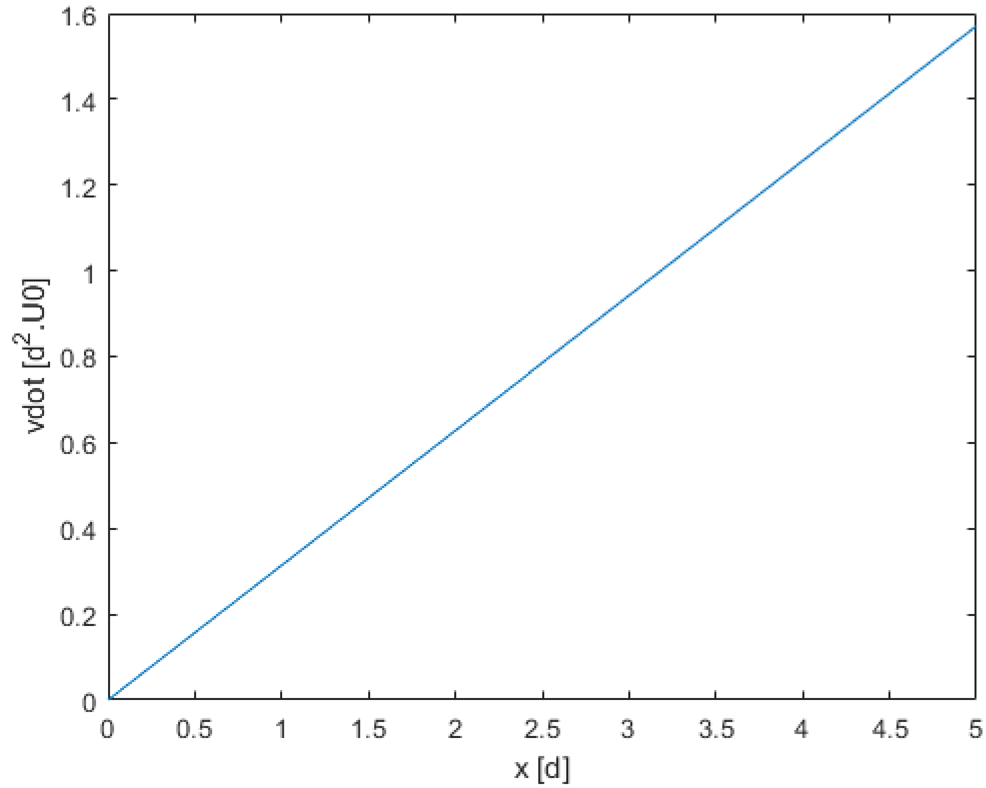
```
u_xdxU0=@(x,r)  (5/x)*exp(-50*r^2/x^2); % a function that returns U/(d*U0) at
 any x and r (used for when U0 and d are not known)
% true u(x,r) given by (5/x)*exp(-50*r^2/x^2)*U0*d

n = 100;
x = 10; % units of [d]
r = linspace(0,5,n); % units of [d]
Uprofile = zeros(1,n);

for i = 1:n
    Uprofile(i) = u_xdxU0(x,r(i)); % units of [(m/s)/(d*U0)] ie non
 dimensional
end

figure
plot(r,Uprofile)
xlabel("r [d]")
ylabel("U [U0]")

clear Uprofile r x n


vdot_xdxU0 =@(x) (1/10)*pi*x; % gives vdot in units of [d^3/s] takes x [d]
% true vdot given by vdot = (1/10)*d*pi*U0*x

mdot_xdxU0 =@(x) vdot_xdxU0(x)*rho;

n = 100;
x = linspace(0,5,n); % units of [d]
vdotprofile = zeros(1,n);

for i = 1:n
    vdotprofile(i) = vdot_xdxU0(x(i)); % units of [d^3]
end
figure
plot(x,vdotprofile)
xlabel("x [d]")
ylabel("vdot [d^2.U0]")

% volumetric flow rate is effectively a velocity times an area, to get
% average velocity over an area we need just divide by that area. Up until
% now we have been integrating by infinite bounds but we now must choose a
% control area lets assume that area in question is the circlular cross
% section of the cone with angle 11.8 degrees at the x we care about
theta = 11.8; % degrees
r = @(x) sind(theta)*x; % [d]
A = @(x) pi*r(x)^2; % [d^2]
aveVelo_xdxU0 = @(x) vdot_xdxU0(x)/A(x); % [U0]
% true aveVelo given by U0*d*vdot_xdxU0(x)/A(x)

aveVelo_xdxU0_10=aveVelo_xdxU0(10);

n = 100;
x = linspace(5,12,n); % units of [d]
```

```
aveVeloprofile = zeros(1,n);
for i = 1:n
    aveVeloprofile(i) = aveVelo_xdxU0(x(i)); % units of [d^3]
end
figure
plot(x,aveVeloprofile)
xlabel("x [d]")
ylabel("Average Velocity [U0]")

clear d rho U0 A0
d = 5;%cm
d = d/100;%m
U0 = 100;%m/s
A0 = (pi/4)*d^2;
[~,~,rho] = joshStdAtm(); % assume rho is at sea level
mdot = @(x) vdot_xdxU0(x)*rho*U0*d;
mdot10d = mdot(10*d);
mdot0 = U0*A0*rho;

% conservation of mass is not broken, the additional mass in the flow is
% accounted for by air which is accelerated by the initial flow.

disp("------------CM2------------")
disp("My work has the following results:")
disp("Mass flow rate [kg/s] at the nozzle is: "+string(mdot0))
disp("Average Velocity [m/s] of flow at 10d is: "+string(aveVelo_xdxU0_10))
disp("Mass flow rate [kg/s] at 10d is: "+string(mdot10d))
disp("there is a linear increase in mass flow rate as distance increases from
 the nozzle, but this is to be expected as jet entrainment will cause nearby
 stationary fluid to become accelerated and enter the jet. this exchange of
 momentum causes the velocity of the jet to drop but increases the area. This
 is only possible in a continuum with a sufficeintly viscous fluid.")
disp("I chose to only plot one half of the velocity profile, but it is
 mirrored across the U axis to create a symetric profile.")
disp("The integrating bounds for all integrals are infinity becuase there is
 not a sharp cuttoff to jet entrainment, rather it decreases radially from the
 jet and asyptotically approaches zero. This means however that theoretically
 at least, any arbitrarily far away particle will have some small change in
 momentum from this jet. For that reason it is logical to integrate over
 indefininte bounds.")
disp("I found vdot by inegrating U(r) over an infinite cylindrical area and
 then found average velocity U-bar by dividing by the crossectional area of
 the cone at the x point of interest.")
```

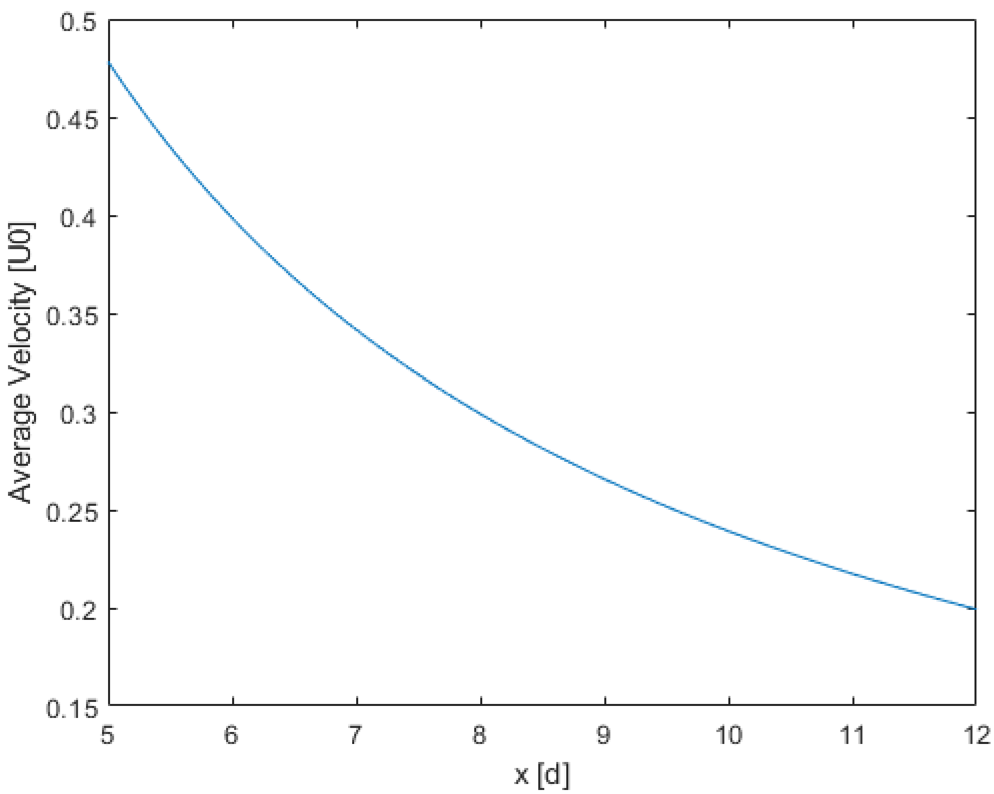
```
------------CM2------------
My work has the following results:
Mass flow rate [kg/s] at the nozzle is: 0.24053
Average Velocity [m/s] of flow at 10d is: 0.23913
Mass flow rate [kg/s] at 10d is: 0.96211
there is a linear increase in mass flow rate as distance increases from the
 nozzle, but this is to be expected as jet entrainment will cause nearby
 stationary fluid to become accelerated and enter the jet. this exchange of
 momentum causes the velocity of the jet to drop but increases the area. This
 is only possible in a continuum with a sufficeintly viscous fluid.
```

I chose to only plot one half of the velocity profile, but it is mirrored across the U axis to create a symetric profile.
The integrating bounds for all integrals are infinity becuase there is not a sharp cuttoff to jet entrainment, rather it decreases radially from the jet and asyptotically approaches zero. This means however that theoretically at least, any arbitrarily far away particle will have some small change in momentum from this jet. For that reason it is logical to integrate over indefininte bounds.
I found vdot by inegrating U(r) over an infinite cylindrical area and then found average velocity U-bar by dividing by the crossectional area of the cone at the x point of interest.





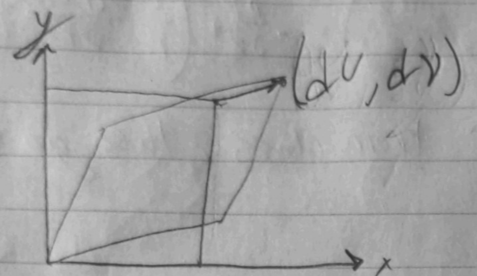
1.6
1.4
1.2
1
0.8
0.6
0.4
0.2
0
vdot [d2.U0]
0
0.5
1
1.5
2
2.5
3
3.5
4
4.5
5
x [d]



0.5
0.45
0.4
0.35
0.3
0.25
0.2
0.15
Average Velocity [U0]
5
6
7
8
9
10
11
12
x [d]

Published with MATLAB® R2022a

FK



We use $\frac{\partial U}{\partial x}$ & $\frac{\partial V}{\partial y}$ to describe the displacement of the corner of the fluid element. This means they can be used to for strain ~~movement~~ assuming strain rate is defined off a normalized deformation

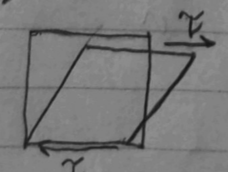
i.e. $\frac{\Delta L}{L} = \delta$. In the x & y dimentions this becomes:

$\delta_x = \frac{\partial u}{\partial x} \Delta x \Delta t$, $\delta_y = \frac{\partial v}{\partial y} \Delta y \Delta t$. However, because fluids deform continuously when subjected to stress, it is more useful to talk about strain rates. This also has the useful advantage of simplifying the above expressions to:

$\varepsilon_x = \frac{\partial u}{\partial x}$, $\varepsilon_y = \frac{\partial v}{\partial x}$, it is easy to imagine

this fluid element in 3-dimensions with the addition of

$\varepsilon_z = \frac{\partial w}{\partial z}$. If the deforming corner moves away from the origin, then the volume of the fluid element will increase. This means we can define the change in volume as $\frac{\Delta V}{V} = \sum_{xyz} \varepsilon$. This gets the definition "Volumetric strain rate" and directly follows from linear strain rate. If the fluid element deforms but does not change volume then there must be 2 forces acting on the fluid element



τ = shear force on fluid element

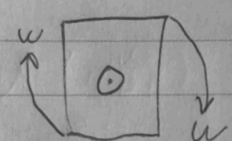
$\varepsilon_{xy} = \frac{1}{2}\left(\frac{\partial u}{\partial y} + \frac{\partial V}{\partial x}\right)$

$\varepsilon_{zx} = \frac{1}{2}\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)$

Fk Page 2

From lecture, we know that vorticity, $\vec{\omega} = \vec{\nabla} \times \vec{V}$ or $\vec{\omega} = \text{Curl}(\vec{V})$

This is simply the nabla vector crossed with the velocity vector and represents a rotation of the fluid about some axis.



CM1

$P_{steam} = \text{Constant}$

1) hydrostatic pressure =
$\rho g h = 1000 \cdot 9.8 \cdot 10m = 98000 Pa$

At ambient T
$P = P_h + P_{amb} = 199325 Pa$

boiling point from lookup table: $T_b = 393.3K$

$T = T_b + 80 = 473.3k$, $\boxed{P = \rho R_{steam} T = \text{constant} = 199325 Pa}$

$\rho = .9108 \ kg/m^3$

$V_{exit} = 15 \frac{m}{s} = \text{constant}$, $U_{steam} = V_e \frac{x}{L}$
where U_s is the boundry between the water and steam,
this creates the ODE $\dot{x} = V_e \frac{x}{L}$.

* See matlab *

note how if ODE starts at $x=0$, It will always remain, but it is an unstable equalibrium, if Pressure increases slightly more, the gyser will go off. I model this with a small but positive x initial

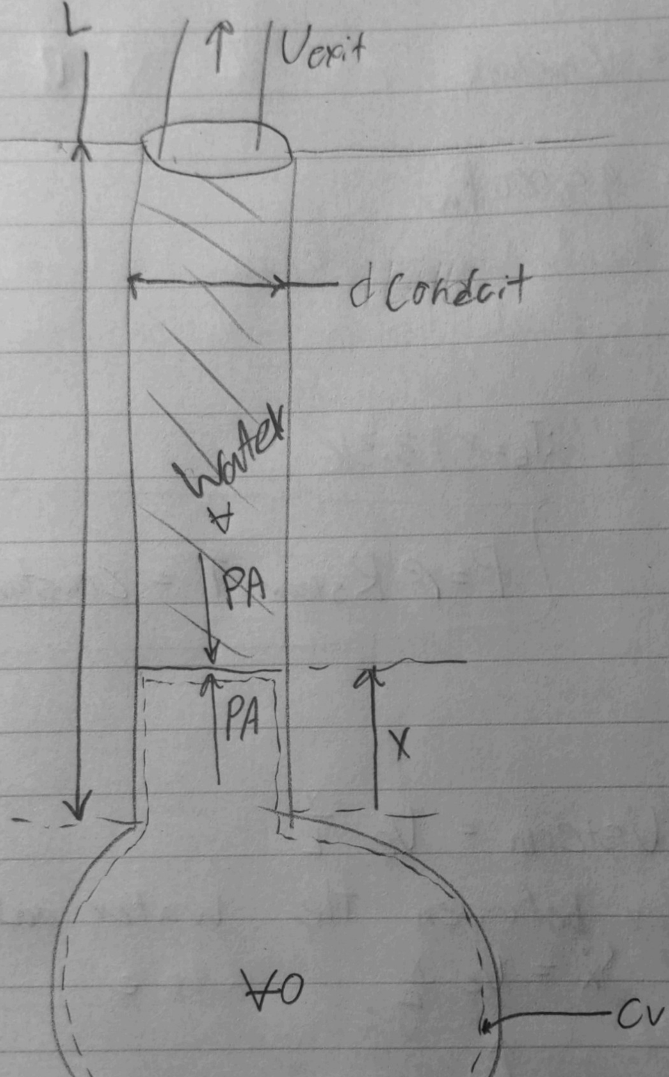
Mass and Volume flow rates are given by $\dot{V} = \dot{x} A$,
where $A = \pi R^2 = \pi \cdot .05^2$.
$\dot{m} = \dot{V} \rho_{water} = 1000 \frac{kg}{m^3} \pi .05^2 m^2 \dot{x}$
where $\dot{x}$ is given from above ode

* Plot in matlab *

ρ of $.75s$ is $\approx .023 \frac{kg}{m^3}$

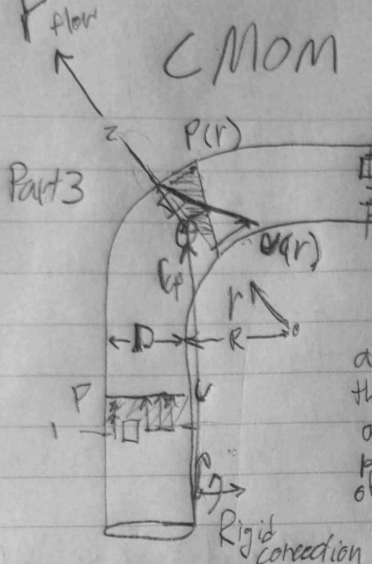
$x(.75s) = 9.2407 \approx 10m$ so the show is almost over after .75s. this is much shorter than a full old faithful show but maybe close to one spurt.

CM1 Page 2



CMom

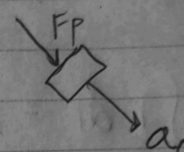
Part3



The bend causes a loss of pressure becase the pocket must be acclerated around the bend by some pressure force, the momentum of the pocket will resist this

Part2

1)  no acceleration

2) 

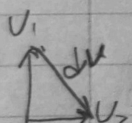
a_c ← centripetal acceleration

3) □ → no accleration

part 2 We are making the assumptions that there is no boundry condition on velocity, ie. this is an inviscid flow and there is a slip condition. This means there will be linear velocity functions.

$$\text{Bend: } F_p = m\,a_c = dP\,\cancel{dA} = \frac{\rho\, d\cancel{A}\, dr\, U(r)^2}{r}$$

$$\int dP = \int \frac{\rho}{r} U(r)^2 dr = P(R) - \int \frac{\rho}{r} U(r)^2 dr = \rho \ln r\, U(r)^2$$



$$F_f = \frac{d}{dt}(mU), \quad F_x = \frac{d}{dt}(m\bar{U}\hat{x}), \quad F_y = \frac{d}{dt}(m\bar{U}\hat{y})$$

$$\dot{m} \quad F_f = \frac{d\vec{U}}{dt} Q\rho \quad \frac{d}{dt} m\bar{U} = \rho Q \bar{U} \quad F_x = \rho Q \bar{U}\hat{x}, \quad F_y = \rho Q \bar{U}\hat{y}$$

$$F_{ball\ bearing} = (-F_x, -F_y)$$

$$M_{ball\ bearing} = r_{centepressure} \times F_f$$

} these forces are created by exchange of momentum between gas and pipe

Part 4) assuming viscous compessible flow, P_e T_e and therefore U_e, should be solved for using isentropic relations.

where $\frac{T_i}{T_e} = 1 + Ma^2 \frac{\gamma - 1}{2}$; $\frac{P_i}{P_e} = \frac{T_i}{T_e}^{\left(\frac{\gamma}{\gamma-1}\right)}$

NSE 2

$$\rho\left(\cancelto{0}{\frac{\partial u_z}{\partial t}} + \cancelto{0}{u_r\frac{\partial u_z}{\partial r}} + \cancelto{0}{\frac{u_\theta}{r}\frac{\partial u_z}{\partial \theta}} + \cancelto{0}{u_z\frac{\partial u_z}{\partial z}}\right) = -\frac{\partial P}{\partial z} + \left(\frac{1}{r}\frac{\partial (r\tau_{rz})}{\partial r} + \cancelto{0}{\frac{1}{r}\frac{\partial \tau_{\theta z}}{\partial \theta}} + \cancelto{0}{\frac{\partial \tau_{zz}}{\partial z}}\right) + \cancelto{0}{\rho g_z}$$

From class, these go to zero

$$\Rightarrow \frac{\partial P}{\partial z} = \frac{1}{r}\frac{\partial(r\tau_{rz})}{\partial r},$$

$$\tau_{rz} = \mu\frac{\partial u_z}{\partial r}$$

See page 2 of NSE

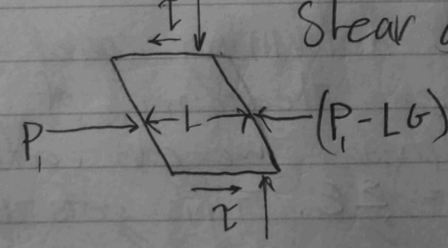
$$\frac{\partial P}{\partial z} = \frac{1}{r}\frac{\partial}{r}\left(\frac{\partial u_z}{\partial r}\right) = G \quad , \quad \frac{\partial P}{\partial z} = G = \text{pressure gradient}$$

Solve for u using PDE

$$u_z = \frac{G}{4\mu}(R^2 - r^2)$$

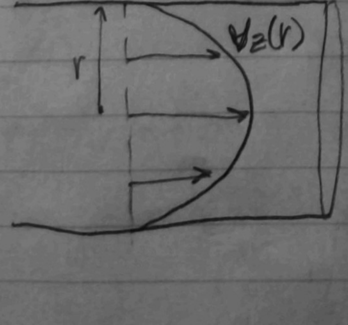
where R is radius of Pipe's, r is position in pipe

Using "no slip" boundry condition, i.e. $u_z(R) = 0$ and the assumption that the profile of velocity is not changing with z i.e. $\frac{\partial u_z}{\partial z} = 0$



Shear acting on a fluid element and pressure acting such that there is a net pressure moving the element in the (+)z direction.

flow velocity in pipe



$$Q = \bar{u}A = \pi R^2 \tfrac{1}{2}u_{max} = \pi R^2 \tfrac{1}{2}\frac{GR^2}{4\mu}$$
$$= \pi R^4 \tfrac{1}{8} G\mu$$

$$Q\rho = \dot{m}$$

NSE PDE

$$\int r \frac{\partial P}{\partial z} dr = \mu \int \frac{\partial}{\partial r}\left(r \frac{u_z}{\partial r}\right) dr$$

$$\frac{\partial P}{\partial z} \int r \, dr = \mu \int \frac{d}{dr}\left(r \frac{\partial u_z}{dr}\right) dr$$

$$c_1 + \frac{\partial P}{\partial z} \frac{r^2}{2} = \mu \left(r \frac{\partial u_z}{\partial r}\right)$$

$$\int \frac{c_1}{r} dr + \int \frac{\partial P}{\partial z} \frac{r}{2} dr = \int \mu \, \partial u_z$$) multiply both sides by ∂r

$$c_1 \int \frac{1}{r} dr + G \int \frac{r}{2} dr = \mu \int d u_z$$

$$c_1 \ln(r) + G r^2 \frac{1}{4} = \mu u_z + c_2$$

from this integral solve c_1 & c_2 using boundry conditions