Orthogonal Cooling hole in a Cross-flow Jet

Elton Freeman

CFD Colloquium

May 18th 2010

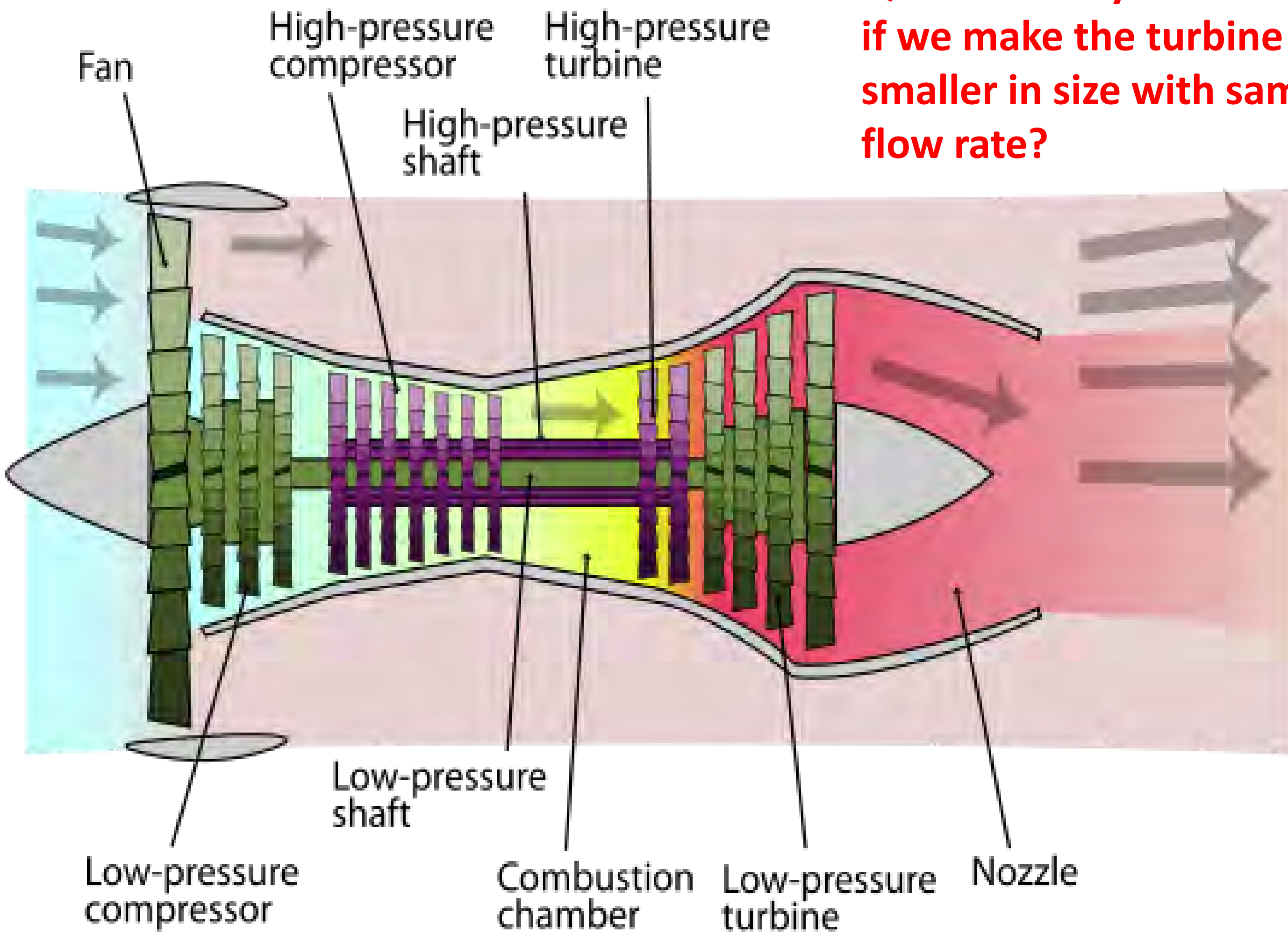
Thin film cooling is one of the newest techniques to improve turbine combustion engine performance to the next level. With advancements in thin film cooling:

A. The efficiency of the turbine engine can be improved by increasing the inlet temperatures.
B. Decrease the volume of the turbine engine, this means of course a smaller fusaloge is needed to house the engine. In most cases this can have other affects on reducing the areodynamics of an airline.
C. With decreases volume, this allows a reduction in the turbines weigth which also means a greater power to weigth ration.

Current disadvantages (Problems with thin film cooling):

A. Thin film cooling only works for cases where the equivalance ratio (mixture from ideal stiochiometeric mixing) is lean, where $\phi < 0.95$ where $\phi = 1.5$ adverse affects are noticed.
B. Air must be bleed from the compressor, that means reducing efficiency from pumping air into the turbine.

Where our current research is being applied, areas including the high-pressure turbine and around the stator areas. Highest pressure and temperature ranges or most prong to failure….



Q1: What do you think happens if we make the turbine engine smaller in size with same mass flow rate?

First engine designs of the 1940's -1950s no special alloys where incorporated, lower operating temperature, material science made the last big jump in turbine performance.

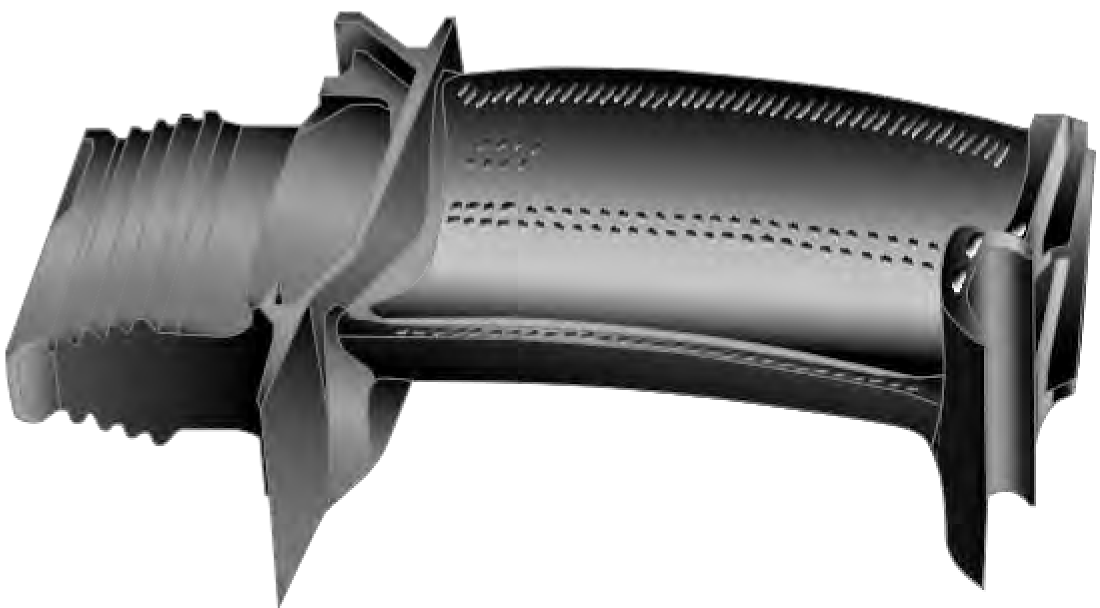
The majority of gas turbines are made from nickel-based alloys. Where nickel-titanium allows have showed strong resistance againist failure of loading at high temps. However current running temperature of the gas turbine (1350°C) is often in excess of the melting point of these Nickel alloys (1200~1315°C)!

TBC has made a slight jump, (Thermal Barrier Coatings) Reduces the Thermal Conductivity at the surface.

Current design challenges in turbine engines, the next step in thin film cooling:

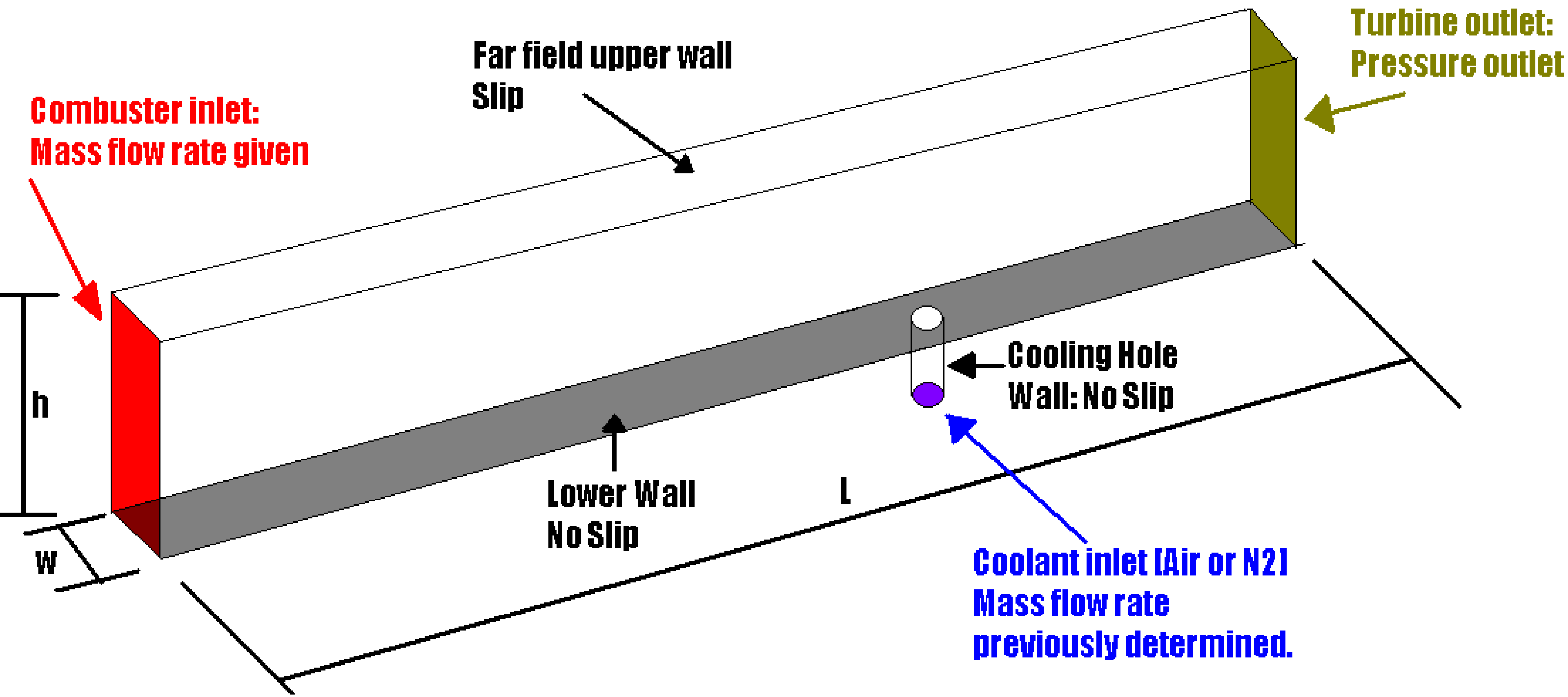
Main idea: Cooling holes allows transport of a cooler, denser gas that floats on top of the turbines surface, seperating the hot exhaust gases from making contact with the blade, and reduction in convection.

Current computational challenges: (i) How much air to inject? (ii) What conditions does thin film cooling work?

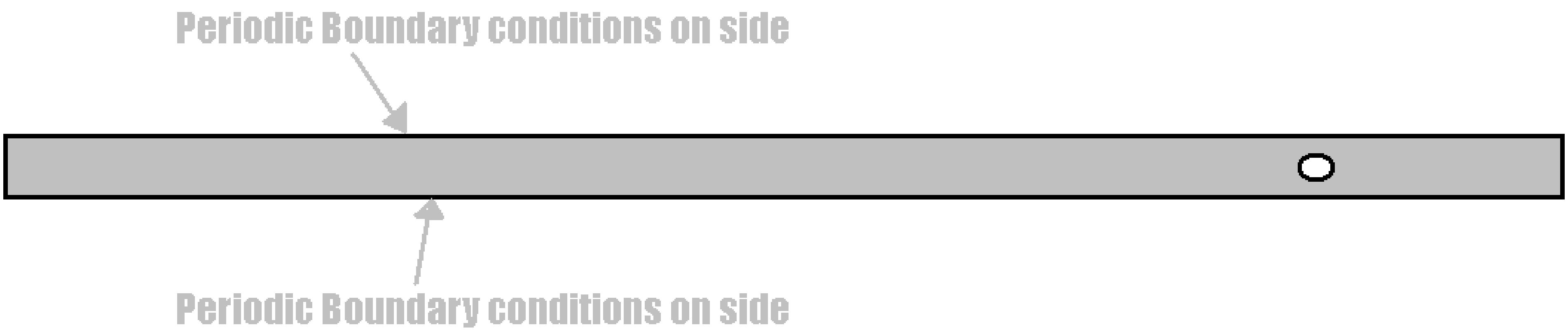


II Geometery Modeled

A. The geometry from the side view:



B. Top view of duct, or plate, showing where the periodic boundary conditions are placed, needed for LES study.



Geometery Specs.				
Description	**Measure**	**units**	**in diam.**	
Total length (x)	0.294	meters	576.471	D
Upstream	0.255	meters	500	D
Downstream	0.041	meters	80.3922	D
Spanwise width (z)	0.00381	meters	7.47059	D
Height (y)	0.0153	meters	30	D
Cooling hole length	0.00255	meters	5	D
Cooling hole Diam.	0.00051	meters	1	D

Flow Specs		
Description	**Measure**	**Units**
Inlet from combustors		
(minimum value)	0.000761694	kg/sec
(Maximum value)	0.003	kg/sec
Inlet Temperature	1836.35	K
Mass Fraction of species:		
C_3H_8	0.0298	
O_2	8.22E-04	
CO_2	0.17	
CO	5.23E-06	
H_2O	0.0598	
Inlet from cooling hole		
(minimum value)	2.40E-06	kg/sec
(Maximum value)		kg/sec
Mass Fraction of species:		
O_2	0.23	

Mesh:

- 7 million hexagonal mesh volumes used.
- Hexagonal is ideal for use with LES.
- Mesh needs to be smooth for LES calculations or gradual course to fine.
- Boundary layer along the wall fine.
- First node off the wall ~3.27×10^{-7} meters, a Y^+ ~ 1.00

To be Turbulent or not to be turbuluent:

Upstream was increased to some 0.255 meters while also adjusting the mass flow rate from the combustors, modeling part throttle and open throttle situations of the turbine.

Picture of turbulent case here from text…….

RaNS calculations exist for the case of a orthogonal jet in cross-flow, we then decided to take it a step further, a LES model…..

Using the LES Model: (Resolved Scales)

We first need to resolve the large eddies for our model, we do this by using a "Filtering Function" G or called (filtering kernal) with seperates the smaller eddies from the larger edies.

$$\bar{u}_i(\vec{x}) = \int G(\vec{x} - \vec{\xi})u(\vec{\xi})d\vec{\xi}, \qquad \textbf{Eqn. 3}$$

Then substituting the filtered values of velocity and pressure we used the "Filtered Navier Stokes Equations"

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu \frac{\partial \bar{u}_i}{\partial x_j}\right) + \frac{1}{\rho}\frac{\partial \tau_{ij}}{\partial x_j}. \qquad \textbf{Eqn. 4}$$

Smagorinsky-Lilly model to model the smaller scales:

$$\tau_{ij} - \frac{1}{3}\tau_{kk}\delta_{ij} = -2\left(C_s\Delta\right)^2 \left|\bar{S}\right| S_{ij} \qquad \text{Eqn. 4}$$

Where the terms arising in the model are:

$$\mu_{sgs} = \rho\left(C_s\Delta\right)^2 \left|\bar{S}\right|$$

Is modeled as the eddy viscosity. The affective viscosity

$$\bar{S} = \sqrt{2S_{ij}S_{ij}}$$

Filtering width is given as

$$\Delta = (\mathrm{Volume})^{\frac{1}{3}}$$

Smaginorsky Constants are: C_s = 0.1 to 0.2

Model for combustion in general:

$$C_3H_8 + 3.5\ O_2 = 3CO + 4H_2O$$
$$2CO + O_2 = 2CO_2$$

$$\text{rate} = k[A]^a[B]^b \quad \text{Eqn 4}$$
$$k = A\exp(-E_A/RT) \quad \text{Eqn 5}$$

1. **We implemented the Arrhenius Rate Equations to describe the reaction process.**
2. **A and B represent the amount of concentrations of species of C_3H_8 and O_2 or CO and O_2.**
3. **K depends on the two species in the reaction, the activation energies and the temperatures.**

Modeling the general combustion with Turbulent Flow:

Previous studies from our group (Cheng-Xian Lin & Richard J. Holder) finds the equivalance ratios above 1.00 (Propane Combustion) shows evidence of a secondary combustion.

Rich mixtures take more time to react, and the concept of reducing the size of the turbine engine means less residence time in the combuster, some fuel doesn't react. Second cause, not enough O_2.

Unburned fuel will therefore burn downstream of the cooling hole, injection of oxygen from cooling hole.

1. Our model uses a simple 2-step propane reaction model to focus on flow mixing and it shows where these secondary reactions will occure and the 2step process is cost efficient.
2. Experimental detials shows that the 2-step combustion process of propane-air is accurate in predicting temperature from the reaction.

Species transport included in this model:

We wanted to model the species mass fractions of propane, di-oxygen, CO and CO_2 before and after the secondary reaction process. The species transport equation that was implemented:

$$\frac{\partial}{\partial t}(\rho Y_i) + \nabla \cdot (\rho \boldsymbol{u} Y_i) = -\nabla \cdot J_i + R_i + S_i \qquad \textbf{Eqn. 1}$$

Where $\mathbf{Y_i}$ is the local mass fraction of a species like propane or di-oxygen. R_i is the species created form the reaction (products in this case). S_i is the rate at which species are created due to dispersion in the flow like coolant in a cross-flow.

Eddy Dissapation Concept modeled:

In order to model the turbulence-chemistry interaction is by use of the Eddy Dissaption Concept (EDC). In order to have secondary combustion, the exhuast inflow is turbulent flow, the incoming coolant O_2 mixes with the turbulent flow eventually leading to the phenomenia of secondary combustion.

Q: How do we describe the the combustion process?

Ans: Its assumed to start at the small turbulent structures or (fine scales) in some time interval τ, but the reaction occurs in some small fraction of both… EDC

$$\zeta^* = C_\zeta\left(\frac{\upsilon\varepsilon}{k^2}\right)^{1/4}$$

$$\tau^* = C_\tau\left(\frac{\upsilon}{\varepsilon}\right)^{1/2}$$

Where C_ζ = 2.1377 and C_τ = 0.4082 called the volume fraction constant and time scale constants, the v is the kinetic viscosity, ε is the turbulent dissaptation term, k is the turbuluent production term.

IV. Numerical computations:

1. Pressure based segerated solver was implemented where the governing equaitons are all solved seperately (e.g. u,v,w,p,T,k,ε) which is memory efficient for this research, but convergence is slow compared to a coupled system.
2. Formulation of implicit was used due to ease of programing, and it is unconditionally stable not matter what time step used Δt.
3. Unsteady time formulation was used w/LES. Flow fields are constantly changing is a good turbulent study.
4. SIMPLEC-collocated grid works best for turbuluent flow.
5. Discretization of second order upstream was used for:
   a. Pressure
   b. C_3H_8 (propane)
   c. O_2
   d. CO_2
   e. CO
   f. H_2O
   g. Momentum discretization used a Bounded Central Differencing Scheme.