PREDICTIONS OF ADIABATIC FILM COOLING EFFECTIVENESS FOR EFFUSION FILM COOLING

Mohammed Albahloul Ahmed Albakai

Department of Mechanical Faculty of Engineering \ Gharyan University

Ahmed Alsiheeri Hassan Laq

Department of Mechanical Faculty of Engineering \ Nalut University

Abstract

This paper investigates innovative cooling techniques for gas turbine combustor walls, crucial for maintaining durability and functionality in high-temperature conditions. Focusing on the combustor liner, the study examines the progression from traditional cooling methods such as louver and splash cooling to advanced techniques like transpiration cooling, which significantly reduces cooling air requirements by up to 50%. Using Computational Fluid Dynamics (CFD) analysis, the research evaluates convection cooling in ducts with 5 mm and 10 mm gap configurations, testing various inlet velocities. The results demonstrate that the narrower 5 mm gap offers a slightly higher cooling efficiency, achieving approximately 70% optimal cooling effectiveness. The study underscores the effectiveness of (CFD) in predicting temperature distributions and optimizing turbine design.

Keywords: Gas Turbine Combustor, Wall Cooling Techniques, Cooling Effectiveness ,Computational Fluid Dynamics (CFD) , Boundary Conditions

تنبؤات فعالية التبريد الغشائي الأدياباتي في التبريد الغشائي الانبعاتي

أ. محمد البهلول أحمد البكاي

قسم الهندسة الميكانيكية كلية الهندسة /جامعة غريان

أ. أحمد السحيري حسن لقع

قسم الهندسة الميكانيكية كلية الهندسة / جامعة نالوت

المستخلص:

يبحث هذا البحث في تقنيات التبريد المبتكرة لجدران غرفة احتراق التوربينات الغازية، والتي تعد ضرورية للحفاظ على المتانة والوظائف في ظروف درجات الحرارة العالية. مع التركيز على بطانة غرفة الاحتراق، تدرس الدراسة التقدم من طرق التبريد التقليدية مثل التبريد بالفتحات والرش إلى تقنيات متقدمة مثل التبريد بالنتح، والتي تقلل بشكل كبير من متطلبات الهواء المبرد بنسبة تصل إلى 50%. باستخدام تحليل ديناميكيات السوائل الحسابية(CFD) ، يقيم البحث التبريد بالحمل الحراري في القنوات ذات تكوينات الفجوة 5 مم و10 مم، واختبار سرعات المدخل المختلفة. توضح النتائج أن الفجوة الضيقة 5 مم توفر كفاءة تبريد أعلى قليلاً، وتحقق فعالية تبريد مثالية بنسبة 70% تقريبًا. تؤكد الدراسة على فعالية ديناميكيات السوائل الحسابية في التنبؤ بتوزيعات درجات الحرارة وتحسين تصميم التوربينات.

الكلمات المفتاحية: غرفة احتراق التوربينات الغازية، تقنيات تبريد الجدران، فعالية التبريد، ديناميكيات الموائع الحسابية (CFD)، الشروط الحدية.

1. Introduction

The functions of the liner are to contain the combustion process and to facilitate the distribution of air to all the various combustor zones in the prescribed amounts. Contemporary liners are typically of brazed and welded sheet metal. The liner must be structurally strong to withstand the buckling load created by the pressure differential across the liner wall. It must also have sufficient thermal resistance to withstand continuous and cyclic high-temperature, oxidant-resistant materials combined with the effective use of cooling air. On modern combustors, up to 50 percent of the total combustor air-mass flow is employed in liner wall cooling. In practice, the liner wall temperature is determined by the balance between the heat it receives via radiation and convection to the annulus air and by radiation to the air casing.

The need to protect solid surfaces exposed to high temperature environments is an old one. In the case of the gas turbine combustor wall, the high temperature environment is gaseous and the last 30 years have witnessed the development of sophisticated cooling techniques. [7-8]

The gas turbine cycle efficiency (η) depends, apart from pressure ratio, on temperature inlet to the turbine blades as shown by the equation below.

$$\boldsymbol{\eta = 1 - \frac{Text}{Tin}} \qquad (1)$$

Where:-

$\eta \; = \; Cycle\ efficiency.$

$T_{ex} = \; Turbine\ exhausts\ temperature.$

$T_{in} \; = \; Combustor\ outlet\ (or\ turbine\ inlet)\ temperature.$

The effect on the combustor wall of increasing T_{in} is such that it leads to the heating of the wall beyond the critical temperature of the wall material. This leads to loss in strength of the material and where there is local heating, buckling of the combustor develops as well as cracks. In short, without adequate protection of the combustor wall, service failure of the flame tube results[2-5].

1.2 Wall-Cooling Technique

Many early gas turbine combustion used a louver technique whereby the liner was fabricated in the form of cylindrical shells that, when assembled, provided a series of annular passages at the shell intersection point. These passages permitted a film of cooling air to be injected along the hot side of the liner wall to provide a protective thermal barrier. The annular-gap heights were maintained by simple wiggle-strip louvers. Air metering was a metering was a major problem with this technique. Splash-cooling devices are much better in this regard. With this system the cooling air enters the liner through a row of small-diameter holes. The air jets impinge on a cooling skirt, which then directs the flow so as to from a film along the inside of the liner wall. However, the machined-ring approach, which features accurately machined holes instead of louvers, combines accurate air flow metering with good mechanical strength and is now widely used. Modern cooling techniques include convection-film cooling, which utilize simple but controlled convection cooling enhanced by roughened walls while providing a protective layer of cool air along the hot side of the wall at each cooling-panel discharge plant. Impingement cooling is well suited to high-temperature poses difficulties in manufacture and repair[1].

The most advanced from of wall cooling now being actively developed is the transpiration system, which has the potential of reducing the required amount of cooling air by as much as 50 percent. With this scheme the cooling air flows through a porous liner wall, first removing heat from the wall itself and then providing a thermal barrier between the wall and the hot combustion gases. [10] The Figure (1) provides a conceptual illustration of the trend, first to double-wall construction where impingement and/or convection techniques increase the heat transfer on the cold side, enhanced by an efflux of film cooling air, and ultimately to transpiration cooling, which is regarded as the ultimate goal. [3-9]

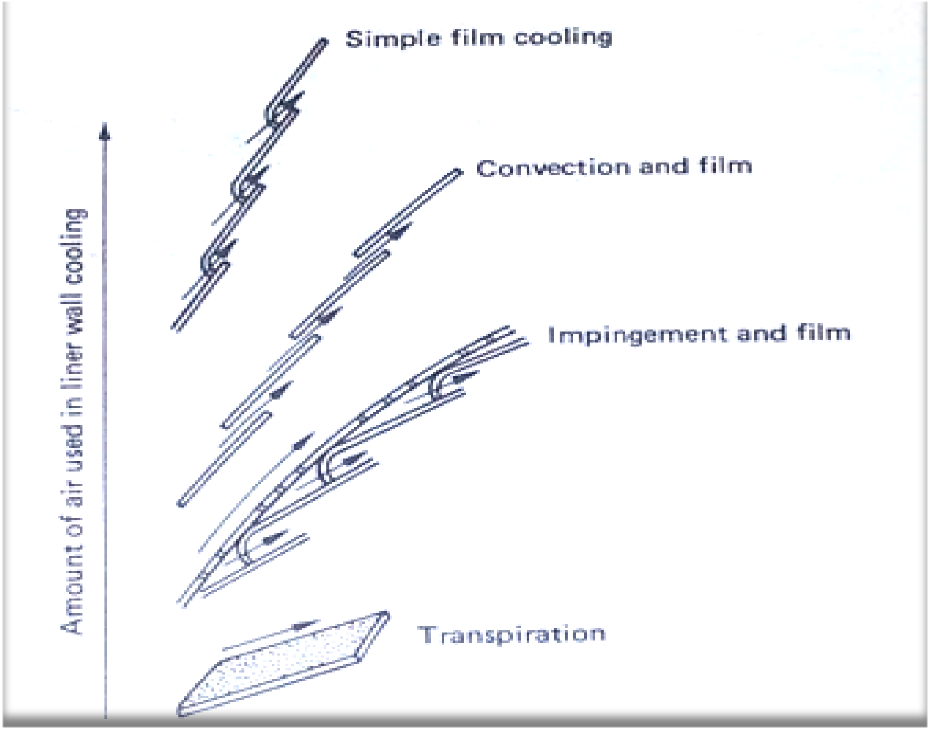


Figure (1) *Trends in combustor wall cooling[1].*

An alternative to increasing the efficiency of cooling techniques is to use protective coating or liner materials that allow operation at higher temperature. Coatings are used to a limited extent in troublesome regions of existing combustion. Candidates for liner materials now under consideration include carbon and carbon composites, ceramic, and alloys of high-temperature materials such as columbium. Techniques for the utilization of these materials are in varying stages of development; none is or routine use in conventional present-day combustors[4].

2. THE COMPUTATIONAL METHODOLOGY

2.1 Introduction to CFD Analysis

Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena by solving numerically the set of governing mathematical equations, Conservation of mass, momentum, energy, species,. [4].

1. The results of CFD analyses are relevant in:-

a. conceptual studies of new designs.

b. detailed product development.

c. troubleshooting.

d. redesign.

2. CFD analysis complements testing and experimentation.

a. Reduces the total effort required in the experiment design and data acquisition.

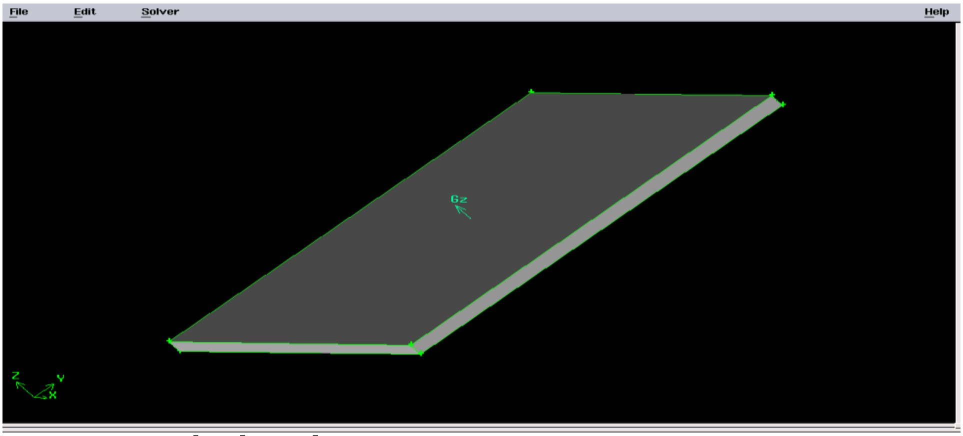
2.2 Physical Model for the Experimental Results of Andrews.

2.2.1 Modeling Of Convection Cooling In A Plain Duct

Procedure

Start GAMBIT.

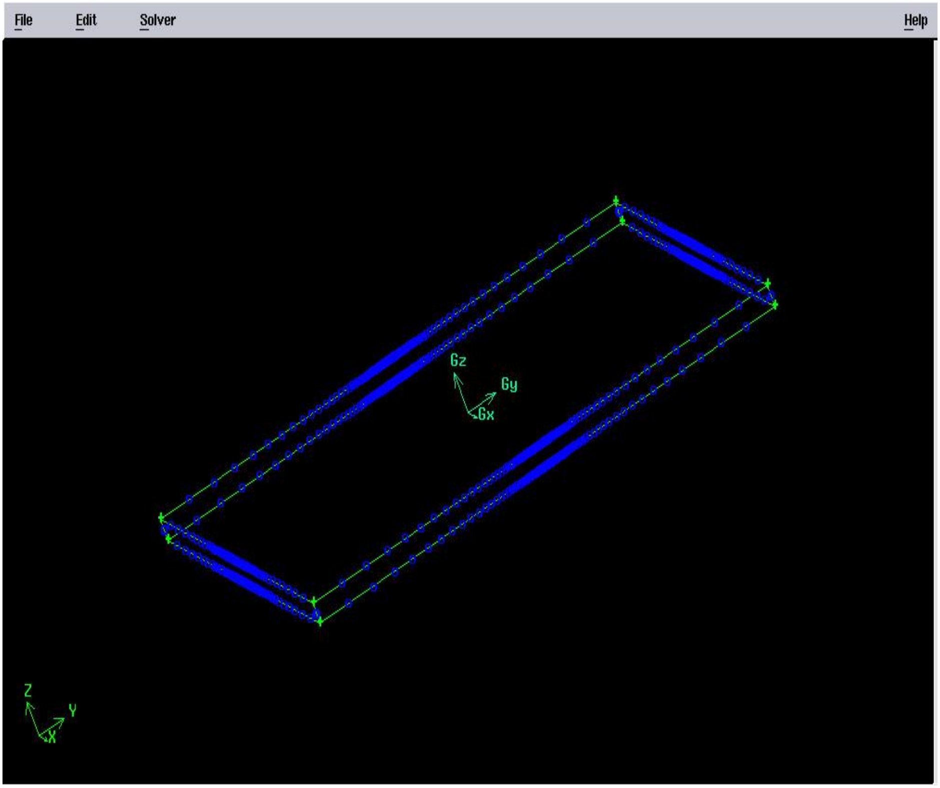
Step 1: Select a Solver



Step 2: Create the duct Plate

Step 3:- Generate an Unstructured Hexahedral Mesh

1. Generate a mesh for one of the small edges.
2. Generate a mesh for one of the small face.



3. Generate a mesh for one of the volume.

Step 4: Set Boundary Types:-

Step 13: Export the Mesh and Save the Session:-

Step 3: Models

Step 4: Materials

Step 5: Operating Conditions

Step 6: Boundary Conditions

Step 7: Solution Controls

Step 8: Initialize

Step 9: Save the case file (case 5mm.cas.gz).

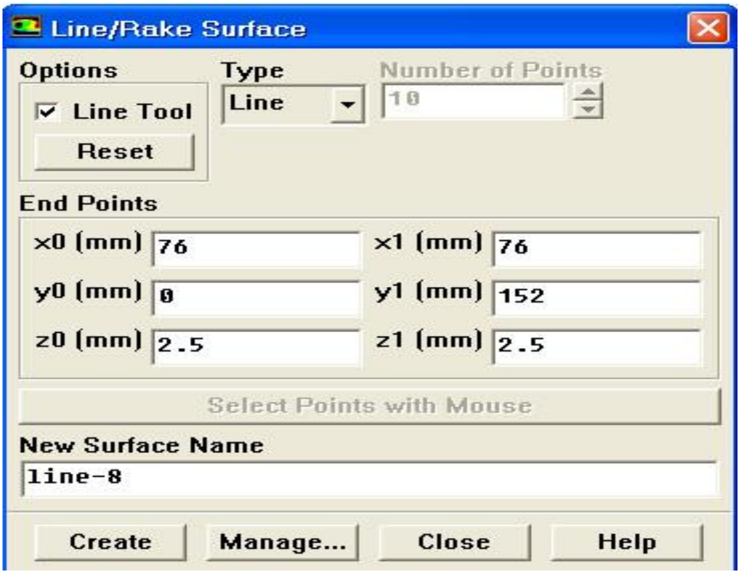
Step 10: Start the calculation by requesting 250 iterations.

Step 11: Save the case and data (case 5mm.cas.gz and case 5mm.dat.gz).

Step 9: Post processing.

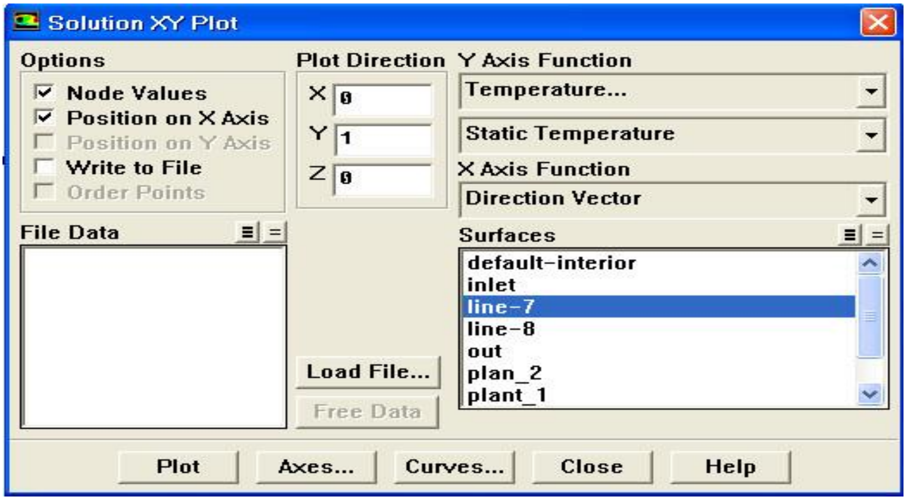
1. Create a line/Rake

Surface □ line/Rake...



1. Create an XY plot of static temperature on the line\Rake created.

Plot □ XY Plot...



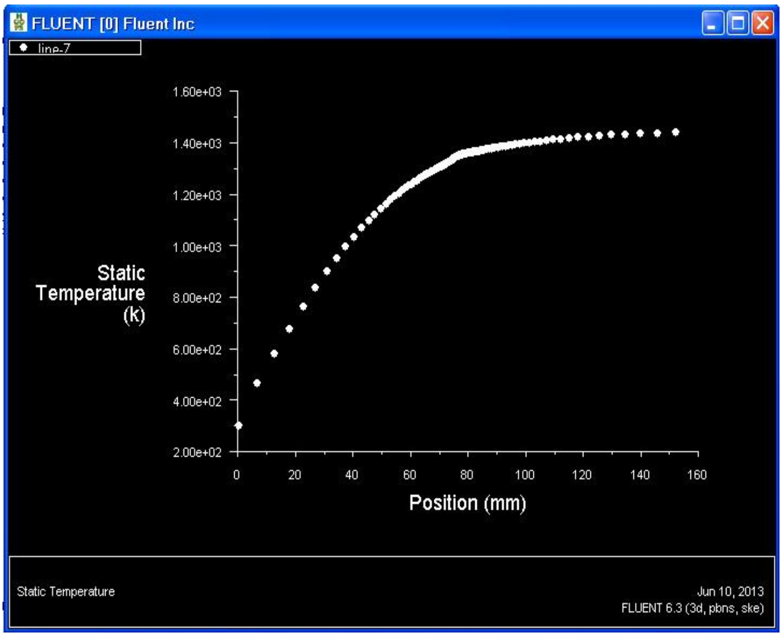
(a) Retain the default Plot Direction.

(b) Select Temperature... and Static Temperature from the y-Axis Function dropdown lists.

(c) Select line-7 selection list.

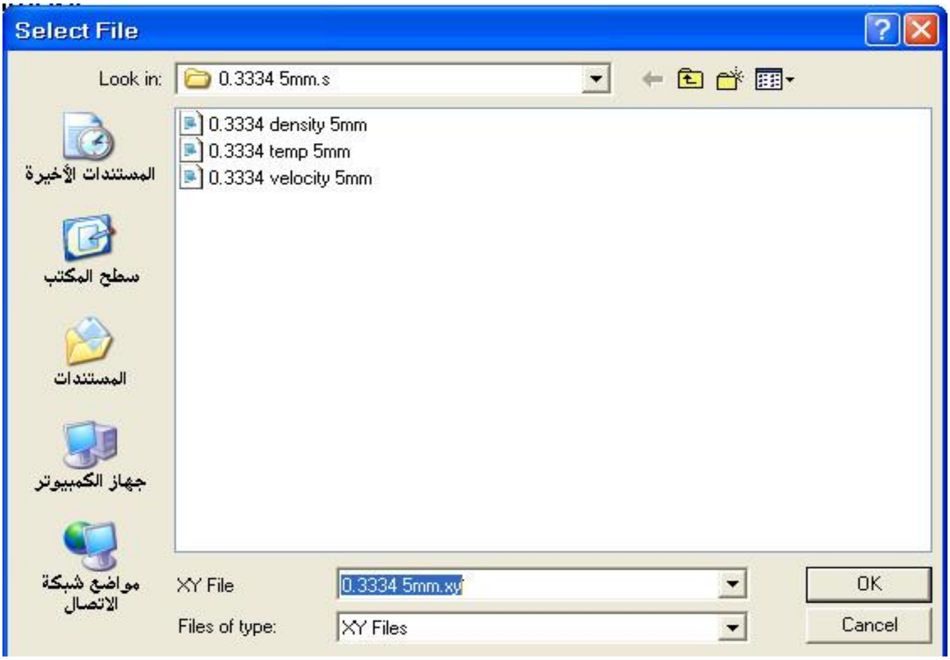
Scroll down using the scroll bar to access line-7.

(d) Click Plot.



(e) Select write to file from solution XY plot

(f) Click to Write……



(g) Click ok.

2.3 Case Study

Parallel plate combustor wall cooling was investigated. The combustor air flowed down the gap between two flat surfaces in a low pressure loss configuration. The work was aimed at combustor liner external air cooling for regenerative combustor cooling prior to entering a lean low NOx combustor. The test rig was, as in Andrews and, 152 mm square and the test case was a duct of 152 mm width and height of 10, and 5 mm with a 152 mm length. Therefore the case study under investigation is similar and the boundary conditions imposed can be varied unlike the previous study of 1700 K for the upper surface temperature and 1200 K for lower one. A duct of 152mm by 152mm and z=5mm and10mm as shown in Figure (2) and Figure (3).

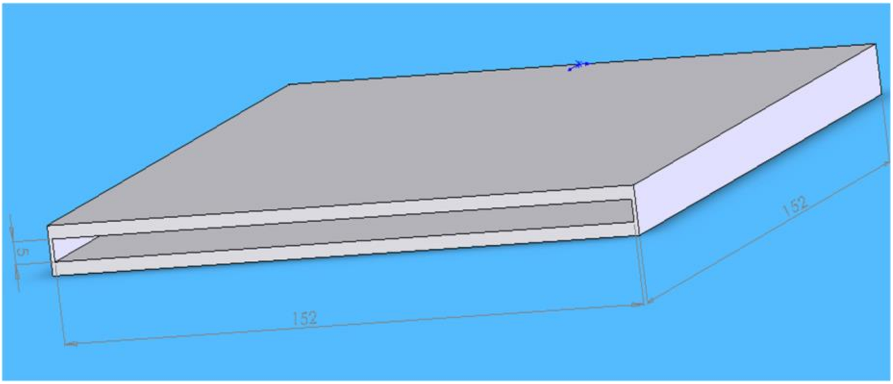


Figure (2) Duct plate with gap 5 mm

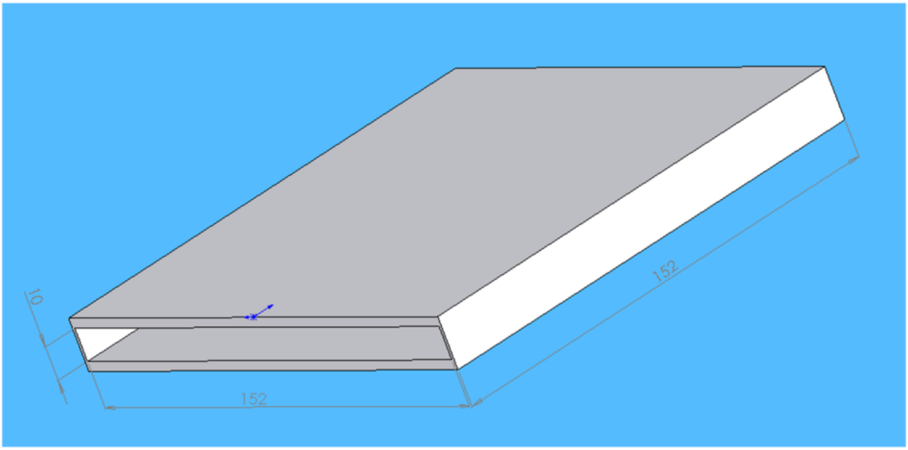


Figure (3) Duct plate with gap 10 mm

Table (1) Case study details.

Case No.	Duct length (mm)	Duct width (mm)	Gap height (z) (mm)
1	152	152	5
2	152	152	10

Table (2) Velocity variation at Z= 5 mm.

Case No.	Duct length(mm)	Duct width(mm)	Gap height (z) (mm)	**Velocity (m/s)**
1	152	152	5	0.3334
2	152	152	5	0.75
3	152	152	5	1.083
4	152	152	5	1.416
5	152	152	5	1.6667

Table (3) Velocity variation at Z= 10 mm.

Case No.	Duct length(mm)	Duct width(mm)	Gap height (z) (mm)	**Velocity (m/s)**
1	152	152	10	0.3334
2	152	152	10	0.75
3	152	152	10	1.083
4	152	152	10	1.416
5	152	152	10	1.667

2.3.1 Wall Temperature Performance

The wall temperature distribution due to cooling for Z=5 mm at different inlet velocities is shown in Figure (4). The slope of the distribution is steeper for lower velocities near the inlet, while get less near the exit of the gap.

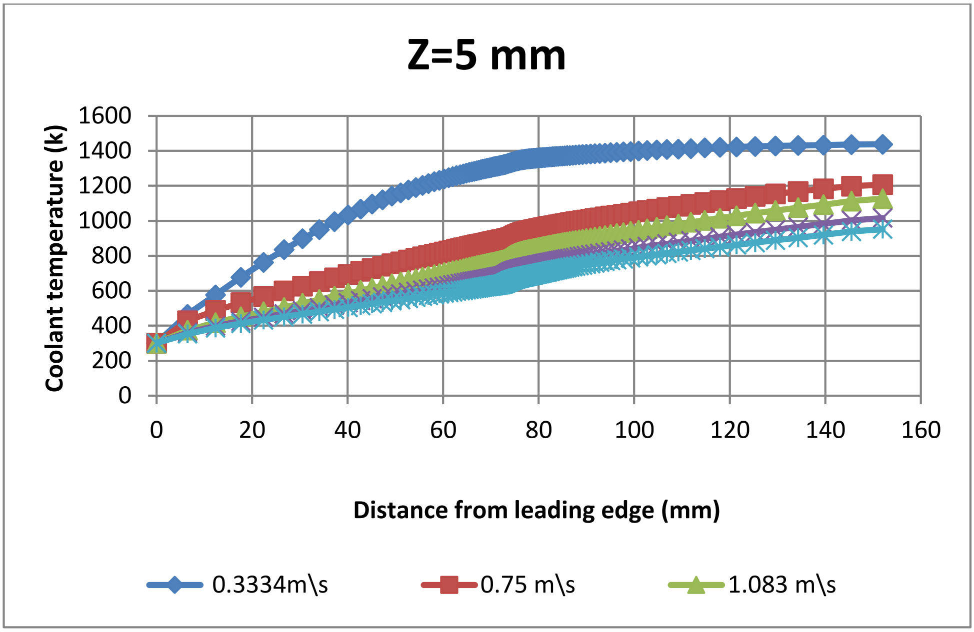


Figure (4) Temperature Distribution for 5 mm Gap.

The maximum temperature out of the gap is 1400 K for the lowest velocity of 0.3334 m/s. This goes to a lower temperature of 880 K for 10mm gap, as shown in Figure (5).

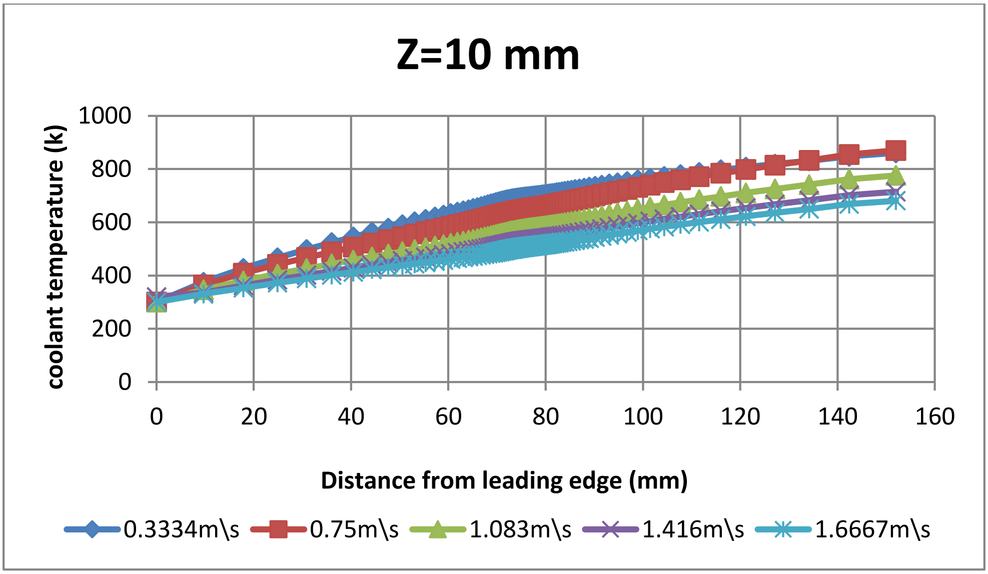


Figure (5) Temperature Distribution for 10mm Gap.

3. Validation of the computational technique of CFD against single rows of film cooling by Andrews.

3.1 Cooling Effectiveness

The cooling effectiveness, defined as :

$$\eta = \frac{T_g - T_w}{T_g - T_c}$$

was calculated for the 5mm and 10mm duct gaps for different coolant mass flow rates G, and the results are shown in Figures (4)and (5). For the 5 mm gap the wall cooling effectiveness was decreased as wall temperature increased for both 5 and 10 mm. From the two graphs the performance of cooling effectiveness in 5 mm is higher than 10 mm gap height. The optimum cooling effectiveness 70% is achieved for both 5 and 10 mm gap height. A cooling effectiveness profile is shown as a function of the distance from leading edge. Decreasing from 1.0 which means 100% cooling effectiveness. The cooling effectiveness decrease as the distance increase up to the trailing edge. The higher the mass flow the higher in cooling effectiveness in general for both 5 and 10 mm.

The current results are compared with the experimental results of [6]. Their data as depicted in Figure (6) shows cooling effectiveness from the distance of leading edge, while the comparison take place in the closest coolant mass flow rate, where the plate size is the same, the value of coolant mass flow rate is about 0.6 for G=1 kg/m^{2}s, the closest results are shown in the middle of the plate. The more close result also shown in the high coolant mass flow rate.

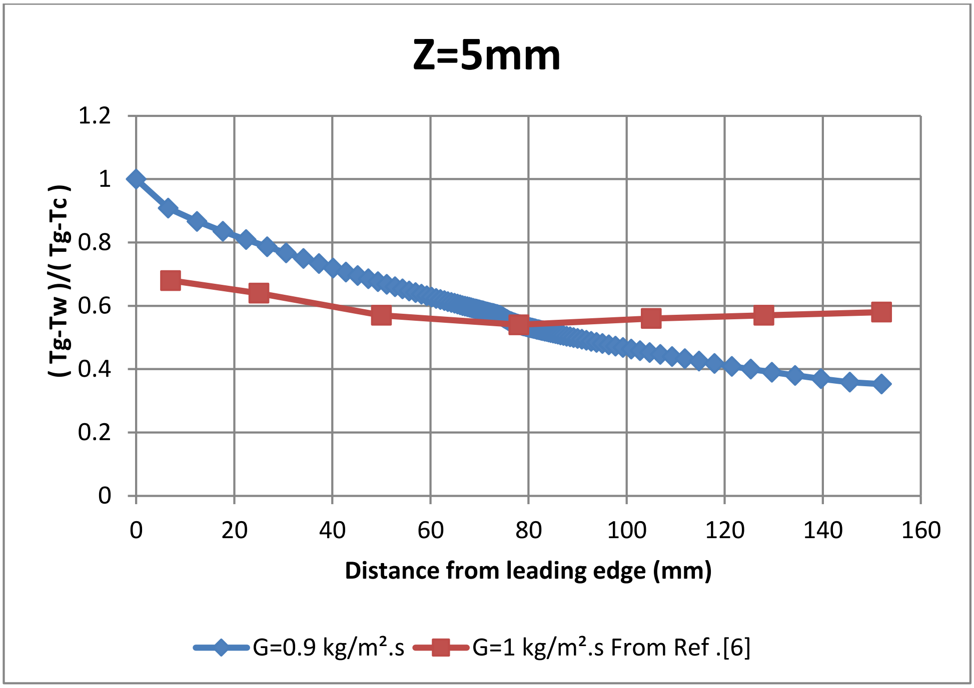


Figure (6) Comparison Between Current Results with experimental results Ref.[6]

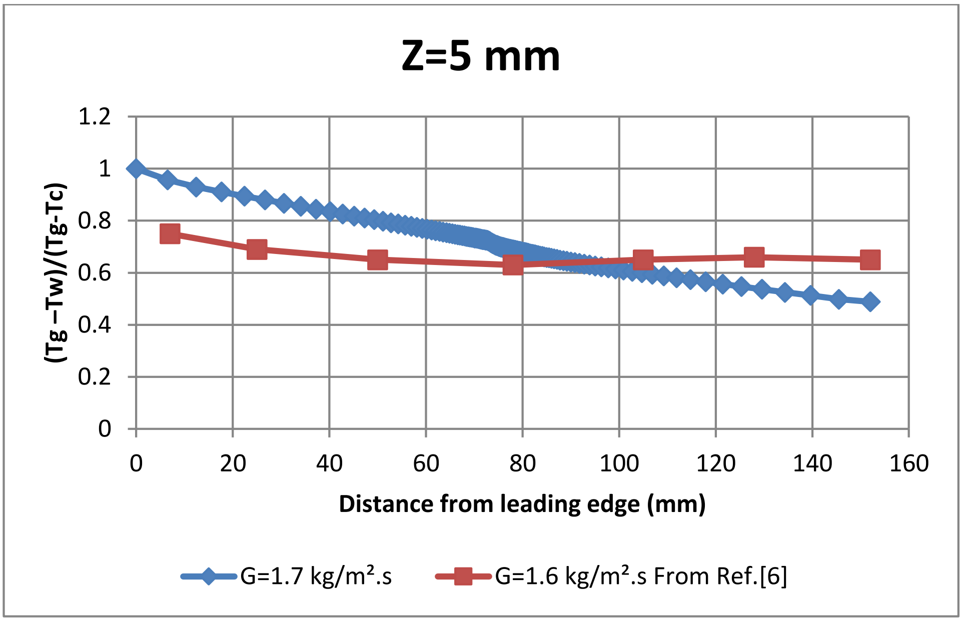


***Figure (7)** Comparison Between Current Results and Ref.[6].*

The same trends for the comparison are shown in Figure (5-6) as the experimental effectiveness is almost constant at 0.7, while the CFD results are going from 1 to 0.5 for G=1.7 kg/m^{2}s.

For the 10 mm gap, the comparison shown in Figures (8) and (9), have better agreement in the slopes of the data and CFD results. For G=1.3 kg/m^{2}s the difference between the CFD and experiments goes between 0.4 to 0.1, while for G=0.9 kg/m^{2}s the difference goes from 0.3 to 0.05.

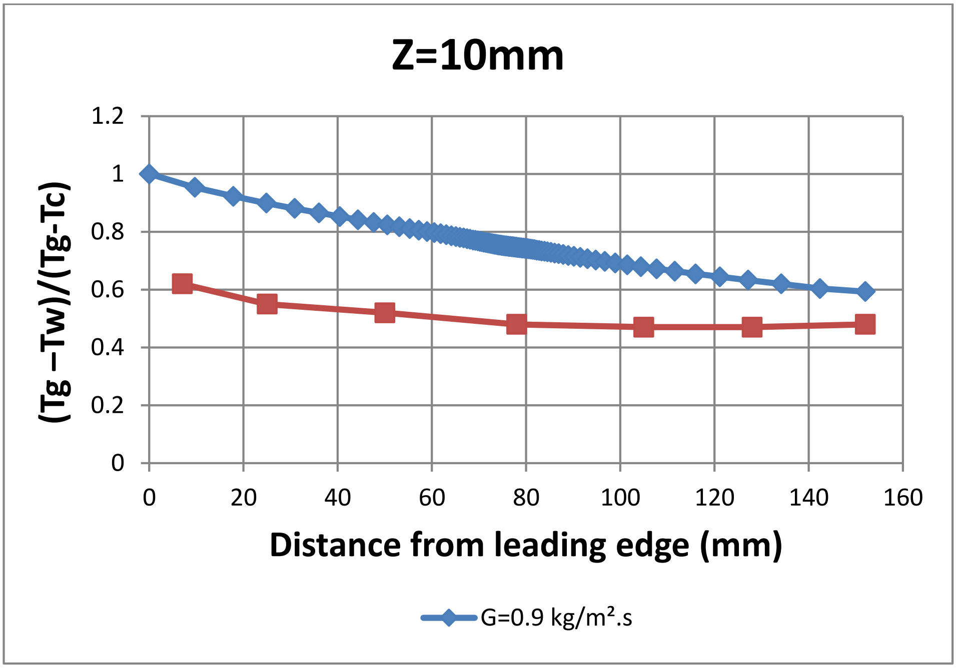


Figure (8) Comparison Between Current Results and Ref.[6].

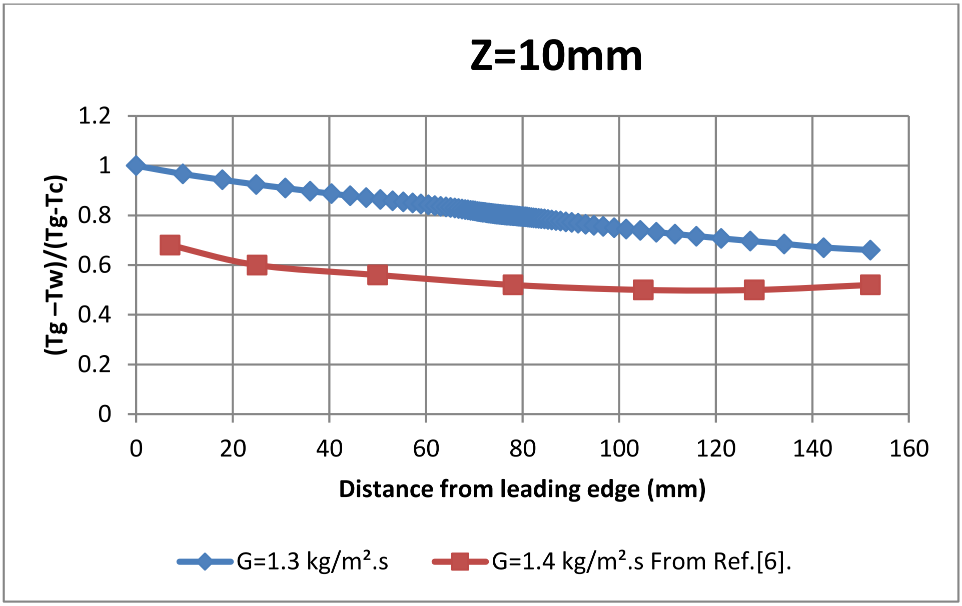


Figure *(9) Comparison Between Current Results and Ref.[6]*

4. Conclusions

From the foregoing study the following can be concluded:

1) The use of the CFD codes like Fluent was helpful in the study of film cooling of the combustor ducts. It proved valuable compared to other types of approaches in recent works which used as based references.

2) Two configuration cases were considered: 5 mm and 10 mm inlet cooling flow gaps.

3) The mesh used in this study consisted of 152 points for the 5 mm gap and 101for the 10 mm in canter line gap experiments. The programmed converged after about 250 iterations for most of the cases. The surface temperature to be cooled were set for 1700 K, while the cooling velocity flow rates ranged from 0.334 m/s to 1.667 m/s, with constant inlet temperature of 300K.

4) The boundary conditions were set for constant-wall temperature and constant inlet flow rates. The κ–ϵ turbulence model, and the surface metal was chosen as steel with conductivity of 16.26 W/m^2.K.

5) The optimum cooling effectiveness 70% was achieved in both two cases of modelling.

6) Comparison with references works shows a reputable agreement either in cooling effectiveness or heat transfer performance.

7) Results of CFD application cooling effectiveness of 5mm is less than 10 mm gap height.

References:

[1] *Fundamentals Of Gas Turbine*, By William W. Bath,1970

[2] *Gas Turbine Theory, By H*. Cohen 4th Edition *,1996* .

[3] *The Jet Engine*, By Rolls Royce-The Jet Engine.5th Edition,1996.

[4] *Mechanics and Thermodynamics Of Propulsion, By Hill Peterson, Third edition, 1970.*

[5] *Gas Turbine Combustor Wall Cooling, By Abraham AwololaAsere, B .Sc.(Hons.), Ass.M.Inst.E., M.Inst.Pet,1986.*

[6] *Planar Duct Heat Transfer With Turbulence Enhancing Obstacles For Combustor Liner External Cooling*, by *Gordon E. Andrews and, GTI, 2007, Canada* .

[7] *Turbine Blade Film Cooling Using PSP Technique*, By Chin Han And Akhilesh P. Rallaband, Texas A&M University, College Station, Texas,2010.

[8] *Enhanced Internal Cooling Of Turbine Blades And Vanes*, By Je-Chin Han And Lesley M. Wright,Texas A&M University College Station, Texas,2009.

[9] *Innovative Cooling Configurations For Low Emission Gas Turbine Combustor*, By Ryan Thomas Hebert B.S., Louisiana State University, August 2004

[10] *Jet Impingement Cooling Configurations For Gas Turbine Combustion* By Eric Ian Esposito B.S., Louisiana State University, August 2006.