

文章编号 : 1000-8055 (2009) 04-0729-11

Isothermal study of effusion cooling flows using a large eddy simulation approach

W. P. Bennett , Z. Yang , J. J. McGuirk

Aeronautical and Automotive Engineering Department ,
Loughborough University , Leicestershire , L E11 3TU , U K.

Abstract : An isothermal numerical study of effusion cooling flow is conducted using a large eddy simulation (LES) approach. Two main types of cooling are considered , namely tangential film cooling and oblique patch effusion cooling.

To represent tangential film cooling , a simplified model of a plane turbulent wall jet along a flat plate in quiescent surrounding fluid is considered. In contrast to a classic turbulent boundary layer flow , the plane turbulent wall jet possesses an outer free shear flow region , an inner near wall region and an interaction region , characterised by substantial levels of turbulent shear stress transport. These shear stress characteristics hold significant implications for RANS modelling , implications that also apply to more complex tangential film cooling flows with non-zero free stream velocities. The LES technique used in the current study provides a satisfactory overall prediction of the plane turbulent wall jet flow , including the initial transition region , and the characteristic separation of the zero turbulent shear stress and zero shear strain locations.

Oblique effusion patch cooling is modelled using a staggered array of 12 rows of effusion holes , drilled at 30° to the flat plate surface. The effusion holes connect two channels separated by the flat plate. Specifically , these comprise of a channel representing the combustion chamber flow and a cooling air supply channel. A difference in pressure between the two channels forces air from the cooling supply side , through the effusion holes , and into the combustion chamber side. Air from successive effusion rows coalesces to form an aerodynamic film between the combustion chamber main flow and the flat plate. In practical applications , this film is used to separate the hot combustion gases from the combustion chamber liner. The numerical model is shown to be capable of accurately predicting the injection , penetration , downstream decay , and coalescence of the effusion jets. In addition , the numerical model captures entrainment of the combustion chamber mainstream flow towards the wall by the presence of the effusion jets. Two contra-rotating vortices , with axes of rotation along the stream-wise direction , are predicted as a result of this entrainment. The presence and characteristics of these vortices are in good agreement with previous published research.

Key words : effusion cooling ; large eddy simulation ; isothermal analysis ; film cooling ;
turbulent wall jet

CLC number : V231.1

Document code : A

Nomenclature

C_s Smagorinsky constant
 D Full coverage effusion cooling hole diameter
 H Full coverage effusion cooling channel height
 i, j, k Cell indices

p Static pressure
 Re Reynolds number
[wall jet : $Re = (u_{inlet})_{max} S / \nu$]
[effusion : $Re = (u_{comb})_{max} H / 2 \nu$]
 s Plane turbulent wall jet case inlet slot height
 $\overline{S_{ij}}$ Mesh resolved strain-rate tensor

t	Time
u_i	Instantaneous velocity Cartesian component
$\bar{u}_i$	Mesh resolved velocity component along i^{th} direction
$\hat{u}_i$	Sub-grid scale velocity component along i^{th} direction
u^+, v^+	Dimensionless inner scaled mean velocity
$(u^+ v^+)^+$	Dimensionless inner scaled turbulence shear stress
V_{jet}	Full coverage effusion cooling jet bulk velocity [average velocity at effusion hole exit plane]
x_i	Cartesian direction component (i^{th} direction)
y^+	Dimensionless surface normal coordinate

Greek Symbols

μ	Dynamic viscosity
ν	Kinematic viscosity
ρ	Density

Subscripts

bulk	Bulk velocity component
comb	Effusion cooling combustion chamber property
i, j, k	Cartesian direction components
inlet	Inlet condition
max	Maximum velocity
supp	Effusion cooling air supply chamber
∞	Free stream property

Introduction

Under routine operating conditions, the flow field inside a modern gas turbine combustion chamber is complex, turbulent and can reach temperatures above 2000 K in the primary combustion zone. Under sustained exposure to these conditions, the combustion chamber liner is susceptible to buckling, from oxidation or high temperature corrosion, or thermal fatigue from differential expansion of the liner material, as described in Lefebvre^[1]. In addition to developments in liner material properties through metal alloys, ceramics and heat resistant coatings, modern combustion chamber designs include aerodynamic manipulation to reduce locally the fluid temperature close to the liner surface.

This paper models a specific class of method for reducing liner temperatures, by injecting cooler fluid from outside the combustion chamber through the liner walls. The resulting film of cooler fluid acts as a thermal barrier, separating the combustion chamber liner from the in-

ternal hot gas. Common strategies used in film cooling include the injection of fluid parallel to the liner surface, through circumferential gaps in the liner, and injecting fluid obliquely through patches of discrete effusion holes.

A common problem with all forms of effusion cooling is the breakdown of the cooling film downstream of the point of injection. This occurs due to turbulent mixing with the surrounding hot gas and through local cross-stream transport of large, high-energy vortices. The increase in fluid temperature with film breakdown causes surface temperature gradients which can introduce local thermal stresses in the liner, contributing to liner fatigue.

This paper focuses on the application of a large eddy simulation (LES) code to film cooling by isolating individual components of practical film cooling designs.

1 Numerical method

The three-dimensional, spatially filtered, incompressible (constant density) form of the mass and momentum conservation equations is given by:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0, \quad (1)$$

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left(2\nu \bar{S}_{ij} - \tau_{ij} \right). \quad (2)$$

Where:

$$\bar{S}_{ij} = \frac{1}{2} \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right), \quad (3)$$

$$\tau_{ij} = \left(\overline{u_i u_j} - \bar{u}_i \bar{u}_j \right). \quad (4)$$

The filtered governing equations are solved in this study using a cell-centred mesh approach. The in-house research code, DELTA, employs a finite volume approach, solving collocated variables using a structured, curvilinear mesh arranged in a multi-block domain decomposition. DELTA is well documented in a number of previous research papers. This includes both its original RANS form, as in Page et al^[2], and as a large eddy simulation CFD code, as in Veloudis et al^[3]. DELTA is capable of computing a wide

variety of incompressible and compressible flow problems. In the current work, the incompressible (constant density) form of DELTA integrates the convective terms in space using a second order central difference method. Pressure-velocity decoupling of the collocated variables is avoided through Rhie-Chow smoothing, with a smoothing factor of 0.1. An explicit time-stepping approach is adopted using a low storage, 3-stage Runge-Kutta method. A constant time step is applied uniformly throughout the computational domain. The Smagorinsky sub-grid-scale (SGS) model is used to approximate τ_{ij} , the turbulent stress component that remains unresolved by the computational mesh. Approaching solid surfaces, near wall damping using the van Driest modification is employed in DELTA. The original form of the Smagorinsky SGS model is given in Smagorinsky^[4]. The van Driest near wall modification originates from van Driest^[5].

2 Numerical model

Two distinct forms of cooling are modelled in this paper. Specifically, a plane turbulent wall jet film cooling model and a full coverage oblique injection effusion cooling model.

2.1 The plane turbulent wall jet

A diagrammatical representation of the plane turbulent wall jet flow field is given in Figure 1.

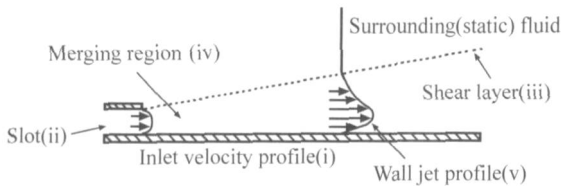


Fig. 1 Representation of the wall jet flow field

This figure represents a two-dimensional (x_1, x_2) plane at a single x_3 location in the computational domain.

Briefly, a jet of air with a plane channel flow velocity profile (i) issues into quiescent air from a slot (ii) and processes downstream along a flat plate. A free shear layer (iii) forms between the wall jet and the quiescent surrounding fluid. This free shear layer interacts with the de-

veloping boundary layer region forming an initial merging region (iv). With increasing downstream distance, the wall jet forms a distinct velocity profile (v) possessing both near wall viscous and free shear layer characteristics.

Validation data for the current plane turbulent wall jet model is provided by the experimental data of Eriksson et al.^[6] and Karlsson et al.^[7]. A numerical comparison is also available through the previous LES study of Dejoan & Leschziner^[8], also based on the measurements of Eriksson et al.^[6]. The current paper uses the same dimensionless geometry as Karlsson et al.^[7] and Eriksson et al.^[6], truncated to $x_1/s = 22$ in the stream-wise direction, as in Dejoan & Leschziner^[8]. An identical Reynolds number of $Re_{inlet} = (u_{inlet})_{max} s/\nu = 9600$ is also defined, where $(u_{inlet})_{max} = 7.34 \text{ m/s}$ is the maximum inlet velocity component in the x_1 direction. $s = 0.02 \text{ m}$ is the jet slot height and ν is the kinematic viscosity for air. A turbulent channel flow velocity profile is defined at the inlet based on a second order extrapolation of the experimental inlet profile given in Eriksson et al.^[6] at the same Re_{inlet} .

The computational domain used in the current work is shown diagrammatically in Figure 2.

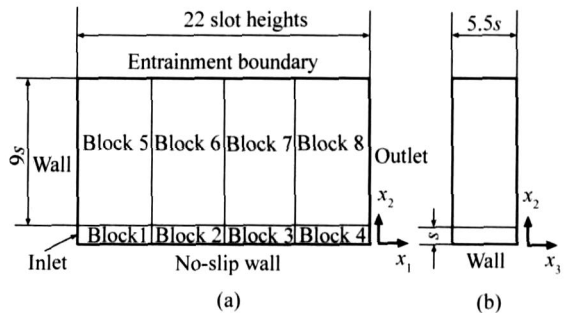


Fig. 2 Wall jet computational domain.

(a) View along x_1 - x_2 plane.

(b) View along x_3 - x_2 plane.

Dashed lines represent multi-block boundaries.

The computational domain dimensions are normalised using the height of the slot used to inject the wall jet, $s = 0.02 \text{ m}$. A distance of 22 slot heights ($22s$) defines the stream-wise length of the computational domain in the x_1 direction. A transverse height of $10s$ defines the height of the computational domain in the x_2 direction.

The computational domain extends 5.5s in the span-wise (x_3) direction. Periodic boundary conditions, used along each span-wise boundary, define an infinite span.

A jet of air issues into the quiescent surrounding air from the inlet boundary with a mean velocity in the x_1 direction. A wall jet develops along the no-slip lower wall and exits the computational domain through the convective outlet boundary. The mean stream-wise velocity profile and turbulence intensity imposed along the inlet boundary are obtained from the measurements of Karlsson et al.^[7] and Eriksson et al^[6]. Specifically, a value of the mean stream-wise velocity is imposed in each computational cell along the inlet boundary through a second-order interpolation of the experimental inlet data at $x_1 = 0$. The mean velocity profile is augmented with a Gaussian random perturbation profile, scaled to match turbulence intensity measurements. The magnitude of the random component is small, less than 1 % of the mean velocity component over the majority of the inlet profile. As stated by Eriksson et al^[6], and Dejoan & Leschziner^[8], the velocity profile at this stream-wise location is effectively laminar with a small temporal fluctuation imposed. In accordance with the approach recommended by Dejoan & Leschziner^[8], an entrainment velocity profile based on a semi-empirical analysis of free jet entrainment (Schlichting^[9]) is imposed along the upper entrainment boundary to facilitate the stream-wise growth of the wall jet.

The computational domain is divided into eight topologically orthogonal blocks. These multi-block boundaries are highlighted in Figure 2 by dashed lines. An equal number of computational cells discretise blocks 1 to 4. Similarly, an equal number of cells discretise blocks 5 to 8. The computational mesh characteristics for the wall jet model are summarized in Table 1.

It should be noted that the present mesh is considerably coarser than the highly resolved LES prediction of Dejoan & Leschziner^[8], who used a grid of 420 (stream-wise) $\times$ 208 (trans-

verse) $\times$ 96 (span-wise) nodes covering the same stream-wise distance of 22 slot heights. Such a fine mesh will rarely be affordable in practice, where using more than 22 slot heights will need to be calculated for the entire combustion film cooled surface. The intention here is to investigate the accuracy that can be achieved with LES on more affordable mesh sizes.

Table 1 Wall jet mesh characteristics

Parameter	Current model
Mesh resolution (stream-wise $\times$ transverse $\times$ span-wise)	105 $\times$ 121 $\times$ 51 nodes
Slot resolution	73 nodes
Height of first cell above wall	$y^+ = 1.693$
Transverse mesh spacing at $x_2 = s$	$\Delta x_2^+ = 2.69$
Transverse mesh spacing at $x_2 = 5s$	$\Delta x_2^+ = 16.80$
Stream-wise mesh spacing at $x_1 = 0$	$\Delta x_1^+ = 10.08$
Stream-wise mesh spacing at $x_1 = 22s$	$\Delta x_1^+ = 18.81$
Span-wise cell spacing	$\Delta x_3^+ = 6.72$

3.2 Full-coverage oblique effusion cooling

A diagrammatical representation of the oblique effusion cooling configuration is shown in Figure 3.

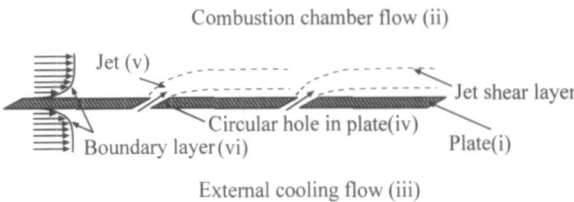


Fig. 3 Representation of the effusion patch flow field

Briefly, a flat plate (i) separates the combustion chamber (ii) and external cooling chamber (iii) flows. The flat plate is pierced by rows of small circular holes (iv) drilled at an oblique angle to the plate. The difference in pressure, maintained between the combustion and external cooling chambers, results in a net mass flux of cooling fluid through each circular effusion hole. A jet (v) of cooling fluid forms at each effusion hole, interacting with the oncoming boundary layer (vi) forming a thermal barrier between the combustion chamber bulk flow and the combus-

tion chamber liner.

The computational domain chosen for the full-coverage effusion cooling model consists of two channels located either side of a 10 mm thick flat plate. Holes of 5 mm diameter are drilled through the plate at 30 ° to the plane of the plate. The spar-wise location of each hole row is staggered with respect to adjacent stream-wise rows , such that each second stream-wise row is in spar-wise alignment. The computational domain for single pair of rows is shown diagrammatically in Figure 4. The complete solution domain for the effusion cooling test case considers a total of 12 rows of holes. The full computational domain is created by merging six of the ‘row-pair ’ computational domain as shown in Figure 4.

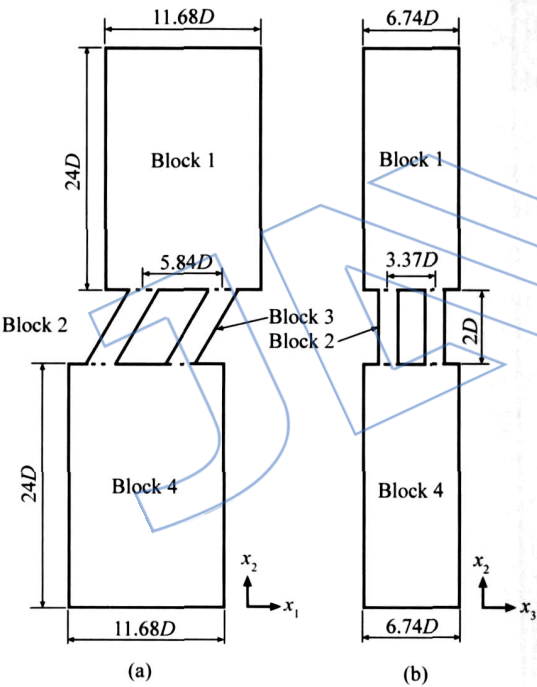


Fig. 4 Oblique effusion cooling computational domain. (a) View along x_1 - x_2 plane. (b) View along x_3 - x_2 plane. First four blocks shown only. Measurements normalized by the effusion hole diameter , $D = 5$ mm

Block 1 in Figure 4 represents the combustion chamber channel. Blocks 2 and 3 represent two effusion hole tubes. Block 4 represents the cooling air supply chamber. Periodic boundary conditions , defined along the spar-wise boundaries of blocks 1 and 4 , define two infinite spar-wise rows of effusion holes. No-slip wall conditions define the upper and lower boundaries in

blocks 1 and 4 , as well as defining the circumference of the effusion tubes in blocks 2 and 3. Locally defined multi-block boundaries connect blocks 1 and 4 with the effusion tubes , blocks 2 and 3. Twelve rows of effusion holes are defined in the current study. To accommodate this situation , the computational domain shown in Figure 4 is repeated a further five times in the x_1 direction. In addition , a further two blocks are defined , aligned with blocks 1 and 4 respectively , in order to allow smooth application of the convective outflow boundary treatment. These two additional blocks have the same dimensions as blocks 1 and 4 in Figure 4. Blocks 1 and 4 connect to the next row of blocks (blocks 5 and 8 respectively) , and so on , using the multi-block boundary condition.

The flow geometry and inlet flow parameters used in the current study are identical to the experimental parameters used by Miron et al.^[10]. The inlet velocity profile , defined in both the combustion chamber and the air supply chamber along the left-hand boundary , is matched to the measurements of Miron et al.^[10]. These geometrical and flow parameters are also used in a reference LES study of a single hole , extrapolated to full-coverage effusion cooling using periodic boundary conditions , by Mendez et al.^[11]. The main flow parameters used in the current study are summarized in Table 2.

Table 2 Oblique effusion cooling flow parameters	
Variable	Value
$Re_{comb} = (u_{1comb})_{max} H/ 2\nu$	17750
$(u_{1comb})_{max}/ (m/ s)$	4. 55
$(u_{1comb})_{bulk}/ (m/ s)$	4. 29
$Re_{supp} = (u_{1supp})_{max} H/ 2\nu$	8900
$(u_{1supp})_{max}/ (m/ s)$	2. 28
$(u_{1supp})_{bulk}/ (m/ s)$	2. 15
$(p_{supp} - p_{comb}) / Pa$	42
Jet bulk velocity $(V_{jet})_{bulk}/ (m/ s)$	5. 6

The computational domain is divided into 26 separate blocks. Using the block numbering

convention in Figure 4, the mesh resolution for each block is given in Table 3.

Table 3 Full-coverage oblique effusion cooling computational mesh resolution

Block	Mesh resolution ($\sum i \times \sum j \times \sum k$)
1, 5, 9, 13, 17, 21 (Combustion chamber blocks)	73 ×61 ×61
2, 3, 6, 7, 10, 11, 14, 15, 18, 19,22, 23 (Effusion tubes)	11 ×29 ×11
4, 8, 12, 16, 20, 24 (Cooling air supply chamber blocks)	73 ×61 ×61
25 (Combustion chamber relaxation block)	51 ×61 ×61
26(Supply chamber relaxation block)	51 ×61 ×61

The orientation of the computational mesh is demonstrated in Figure 5 , which shows a section of the computational mesh for the first four blocks , as defined diagrammatically in Figure 4. The mass flux crossing the outlet boundary of blocks 25 and 26 is adjusted to ensure a continuous supply of cooling air , of the correct magnitude , passes through the effusion tubes as the computation progresses.

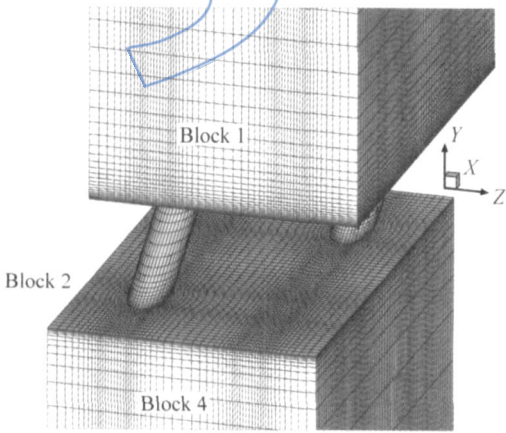


Fig. 5 Computational mesh for blocks 1 to 4

3 Results and discussion

3.1 The plane turbulent wall jet

A simplified model of tangential film cooling is presented in this paper using the case of a plane turbulent wall jet in quiescent air under i-

sothermal conditions. Although of simple geometry , a number of complex boundary layer/free shear flow interactions are reported for the plane turbulent wall jet case. The interaction of the outer free shear region with the inner boundary layer region in a plane turbulent wall jet is documented in a number of studies , including Launder & Rodi ^[12-13] , Eriksson et al. ^[6] , George et al. ^[14] , Barenblatt et al. ^[15] , Dejoan & Leschziner^[8] and Dejoan , Wang & Leschziner^[16] .

Downstream of $x/s = 0$, the wall jet undergoes development from the essentially laminar channel flow profile to a fully developed turbulent wall jet. As reported in Eriksson et al. ^[6] , transition initially occurs in the outer shear layer region , as indicated by a negative mean transverse velocity in the outer region at $x_1/s = 5$. Further downstream , by $x_1/s = 10$, transition of the near wall region has started. At $x_1/s = 20$, the start of a fully developed near wall region is measured by Eriksson et al. ^[6] , and merging of the viscous dominated inner region with the free shear outer region is complete. By the next station , $x_1/s = 40$, measurements indicate that transition of the wall jet is virtually complete. In order to reduce the computational demand of the LES prediction , and to provide a close comparison with the highly resolved LES results of Dejoan & Leschziner^[8] , the computational domain is restricted to a total length of $x_1/s = 22$. Justification for this truncation comes from the observation that the measurements of Eriksson et al. ^[6] , appropriately normalized , are broadly self-similar downstream of $x_1/s = 20$, as reported in Dejoan & Leschziner^[8] .

The downstream decay of the maximum velocity is given for the present results in Figure 6. Overall , the current prediction provides a close match to the measurements of Eriksson et al. ^[6] . The maximum velocity is , however , over-predicted for $x_1/s > 5$. This over-prediction is also reported in the highly resolved LES results of Dejoan & Leschziner^[8] , indicating a higher stream-wise momentum flux and slight under-prediction of the wall shear stress in the

present computation.

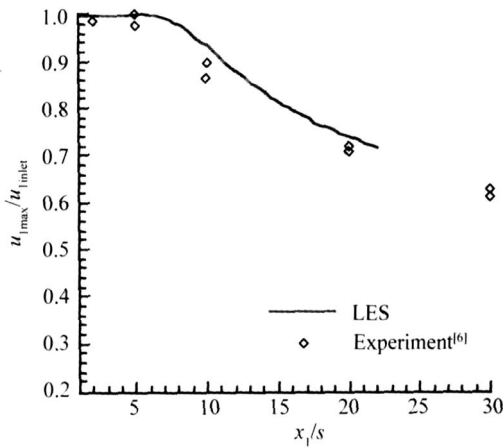


Fig. 6 Downstream development of the wall jet maximum velocity

The over-prediction of the stream-wise momentum flux is also evident in the mean stream-wise velocity profile at $x_1/s = 20$, as given in Figure 7. This figure is scaled using viscous wall properties, defined as inner scaling. Figure 7 also highlights a number of important differences in the wall jet development from that of a classic turbulent boundary layer. The near wall region of the wall jet in Figure 7 closely approximates $u^+ = y^+$ (dashed line) over the region $y^+ < 5$. Above $y^+ = 5$, the wall jet and classic viscous sublayer profiles diverge. The viscous sublayer region of the wall jet velocity profile is smaller than that of a classic turbulent boundary layer, which starts to diverge from the $u^+ = y^+$ profile at $y^+ = 8 - 10$. The reduced viscous sublayer height predicted in the LES results matches closely the measurements of Eriksson et al.^[6] and the highly resolved LES results of Dejoan & Leschziner^[8].

Further from the surface, the height over which the measurements (square symbols) follow the log-law approximation (dash-dot line) is restricted to the region $30 < y^+ < 80$. This log law range is reported in Eriksson et al.^[6], and in Dejoan & Leschziner^[8], who also report an over-prediction of the velocity maximum for the highly resolved LES study.

The mean transverse velocity profile at $x_1/s = 20$ is given in Figure 8, normalized using inner scaling properties. A good match of the

LES data with the measurements of Eriksson et al.^[6] is evident. The absence of a negative velocity in the inner region indicates that transition of the near wall region is complete. A negative transverse velocity occurs above $y^+ = 1000$ as fluid is entrained from the surrounding quiescent fluid into the wall jet outer region.

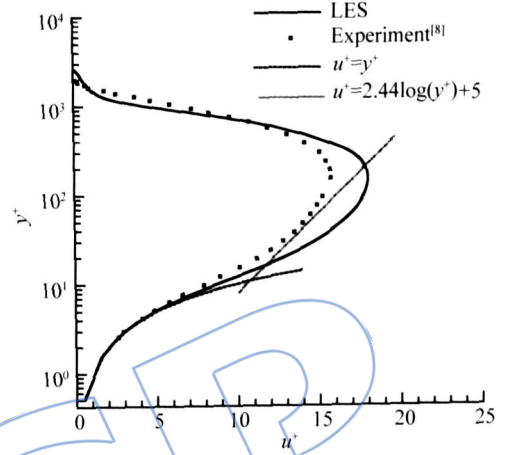


Fig. 7 Wall jet stream-wise velocity at $x_1 = 20s$

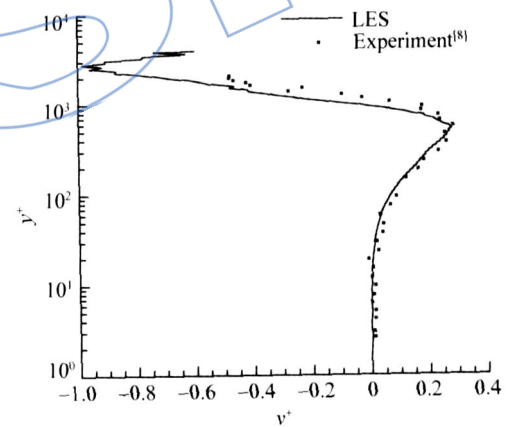


Fig. 8 Wall jet transverse velocity at $x_1 = 20s$

A well-known characteristic of the plane turbulent wall jet is the difference in the height at which the turbulent shear stress and turbulent shear strain are zero. This shift of the zero shear stress and zero shear strain location is evident by comparing the mean stream-wise velocity profile in Figure 7 with the turbulent shear stress profile given in Figure 9.

The rate of turbulence production by shear strain is widely reported to be zero in the region of the mean stream-wise velocity maximum. The velocity maximum in the LES results of Figure 7 occur at $y^+ = 165$. In contrast, the height at

which the turbulence shear stress crosses zero in Figure 9 is $y^+ = 118$, marginally lower than the measurements of Eriksson et al.^[6]. As reported in Dejoan & Leschziner^[8], the dislocation of the zero shear stress ($y^+ = 118$) and zero shear strain ($y^+ = 165$) implies large scale transport of the turbulent shear stress in the interaction region, located between the wall jet inner and outer regions.

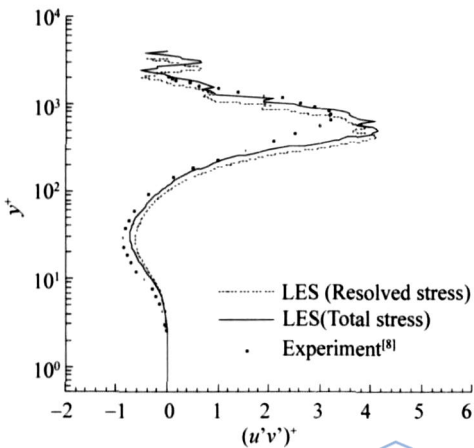


Fig. 9 Wall jet turbulent shear stress at $x_1 = 20s$

The LES results in Figure 9 demonstrate a good overall match of the turbulent shear stress with the measurements of Eriksson et al.^[6] over the majority of the wall jet height. The contribution to the total turbulent shear stress (mesh resolved and sub-grid-scale stress) from the sub-grid-scale component is small and, as expected, is confined to the near wall region ($y^+ < 80$).

In addition to the dislocation of the zero turbulent shear stress and shear strain locations, larger turbulent length scales are reported further towards the inner region, when comparing the plane turbulent wall jet to the classic turbulent boundary layer. In terms of turbulence modelling, this has a significant impact on the range of methods that are potentially able to accurately predict the turbulent wall jet. Dejoan & Leschziner^[8] and Dejoan, Wang & Leschziner^[16] conclude that a turbulence model must account for turbulent shear stress transport in the interaction region to model the turbulent wall jet development accurately. Even RANS turbulence models based on the Reynolds stress transport equation indicate a considerably inferior prediction

compared to LES predictions, as reported in Dejoan, Wang & Leschziner^[16].

Launder & Rodi^[12-13] report that increasing the velocity of the flow outside of the wall jet causes the wall jet inner region to increasingly resemble the classic turbulent boundary layer profile. The wall jet case represents, therefore, a sound basis with which to test an LES code before progression to other jet/main flow velocity ratios and more complex geometries.

3.2 Full-coverage oblique effusion cooling

As with the tangential film cooling analysis, a simplified model of oblique effusion patch cooling is considered using staggered rows of effusion holes drilled at 30° to a flat plate. An example of the instantaneous velocity is given in Figure 10 for the region covering effusion rows seven to twelve.

Figure 10 shows a single iso-surface level equal to $1.05V_{jet}$, where V_{jet} is the bulk velocity in the effusion tube. Grey planes define the upper and lower surfaces of the flat plate. Ellipses define the radial extent of each hole on the plate upper and lower surfaces. Figure 10 highlights the penetration of each jet into the combustion chamber mainstream flow, as well as the breakdown of each jet into small-scale turbulence and the downstream coalescence of successive jets along the same span-wise alignment.

The stream-wise development and coalescence of the jets, highlighted in Figure 10, can be analysed further through consideration of the change in the distribution of stream-wise velocity across each effusion hole. The mean stream-wise velocity at four stream-wise locations across effusion row nine, of the twelve row effusion case, is given in Figure 11. The mean stream-wise velocity profile three diameters downstream of the ninth row is shown in isolation in Figure 12, for greater clarity.

Identification of the main features in Figures 11 and 12, labelled (i) - (iii), and analysis of their downstream development is possible. The maximum stream-wise velocity for profiles (b) - (d) in Figure 11 is labelled (i). This local peak

shows the location and magnitude of the jet injected at the ninth effusion hole row , i. e. in the location of profile (b) in Figure 11.

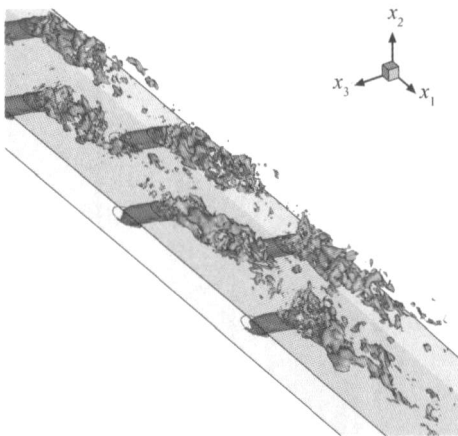


Fig. 10 Instantaneous total velocity across rows 7 - 12. Single iso-surface at $1.05V_{jet}$. Grey planes define the upper and lower surfaces of the plate

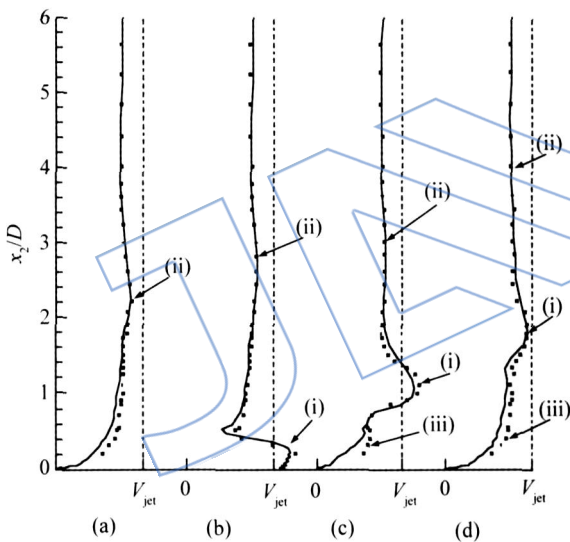


Fig. 11 Mean stream-wise velocity profiles across the 9th stream-wise effusion row. Profiles located at (a) 3 hole diameters upstream of the hole centre , (b) directly above the hole centre , (c) 3 hole diameters downstream of the hole centre , (d) 6 hole diameters downstream of the hole centre. Solid line : L E S prediction , Square symbols : measurements of Miron^[17] , dashed line : effusion tube bulk velocity (V_{jet}) used as reference for profiles (a) - (d) . Location of $u_1 = 0$ also explicitly labelled for profiles (b) - (d)

The transverse penetration of the jet into the combustion chamber flow and the downstream decay of the stream-wise velocity component is evident by tracking this feature over pro-

files (b) - (d) in Figure 11. By extension , the flow feature labelled (ii) is the mean stream-wise velocity contribution from the jet injected at row seven. This feature is more clearly defined in Figure 12 due to the aspect ratio used in plotting this figure. A coalescence of the two jet contributions is evident in Figure 11 (c) - (d) . Downstream of the station used in Figure 11 (d) , a further effusion hole row injects cooling fluid into the combustion chamber. In terms of the mean stream-wise velocity profile therefore , the region above (i) represents a coalescence of jets from all upstream effusion hole rows along the same span-wise alignment. The contribution of the upstream jets gradually diminishes as the jets decay and spread with increasing stream-wise distance.

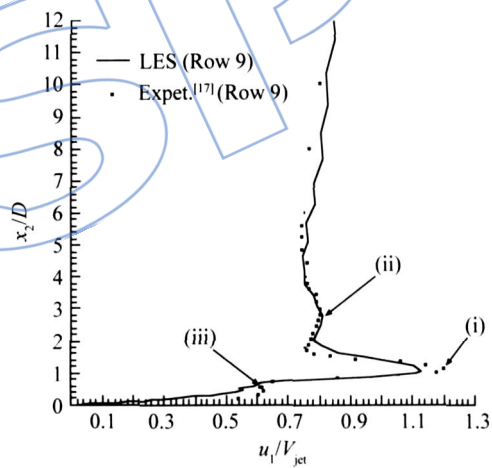


Fig. 12 Development of the stream-wise velocity at 3 hole diameters downstream of row 9

A further local velocity peak is evident in Figure 11 (c) - (d) and Figure 12 below the location of the jet labelled (i) . This local peak is labelled (iii) in Figures 11 and 12 and is a secondary effect resulting from the presence of the primary jet (i) . This flow feature is analysed in a number of published numerical and experiment research studies , including Yavuzkurt et al.^[18] and Tyagi & Acharya^[19] . The local maximum (iii) is caused by entrainment of the main combustion chamber flow towards the wall by the transverse rotation of the effusion jet. This entrainment , which is captured in the current L E S prediction , is shown in Figure 13.

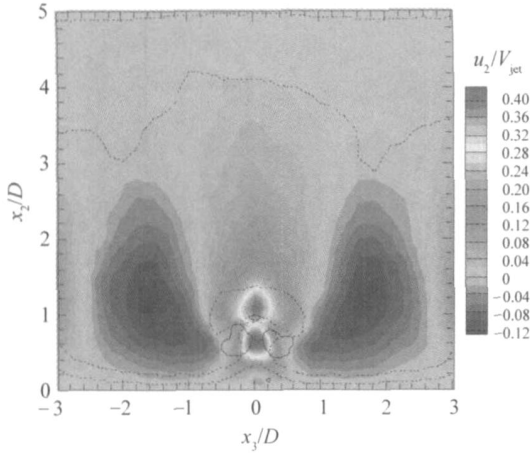


Fig. 13 Span-wise distribution of the mean stream-wise and transverse velocities at 3 effusion hole diameters downstream of row 9. Flood colour plot: Transverse velocity. Dotted black line: Iso-contours of stream-wise velocity, $u_{1\min} = 0.5V_{\text{jet}}$, $u_{1\max} = V_{\text{jet}}$, $\Delta u_1 = 0.25V_{\text{jet}}$. Solid black line: Q -criterion iso-contour at $Q = 2V_{\text{jet}}^2/D^2$. Span-wise distance referenced to hole centre at row 9

The Q -criterion iso-contours in Figure 13 highlight the centre of two contra-rotating vortices, with axes of rotation along the main flow direction. These are shown in Figure 13 as solid black lines. The Q -criterion, or Laplacian of pressure, is defined as:

$$Q = \frac{1}{2} (\Omega_{ij} \Omega_{ij} - S_{ij} S_{ij}) \quad (5)$$

where:

$$\Omega_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \right) \quad (6)$$

$$S_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (7)$$

Further details and application of the Q -criterion are given in Dubief & Delcayre^[20], Tyagi & Acharya^[19] and Luo et al.^[21].

A negative mean transverse velocity highlights an entrainment of the main flow towards the wall. This is evident in Figure 13, at the outer edges of the two contra-rotating vortices, as dark grey regions. A positive transverse velocity, between the two contra-rotating vortices, moves fluid away from the wall towards the main flow region. This region of positive transverse velocity is highlighted by two small regions of grey contours surrounded by white contour rings at $x_3/D = 0$.

With regard to the current LES results, a slight under-prediction of the maximum stream-wise velocity labelled (i) in Figures 11 and 12 is evident, along with a slight over prediction in the transverse spreading of the jet in this region. A slight under-prediction of the fluid entrained by the effusion jet is also evident at the location labelled (iii) in Figures 11 and 12. Overall, however, the match of the LES prediction to the measurements of Miron^[17] is good.

4 Concluding remarks

Based on the results presented in this paper, the following conclusions may be drawn:

1) Two isothermal models have been developed to study two different methods of film cooling typically found in gas turbine combustion chamber designs. The current LES method is capable of accurately reproducing the essential flow physics of an isothermal plane turbulent wall jet and oblique effusion patch cooling.

2) Transition of an isothermal plane wall jet to fully developed turbulent behaviour in both the outer free shear layer region and the inner boundary layer region is reproduced by the current LES model. The separation of the zero turbulent shear stress and zero turbulent shear strain locations, which indicate significant turbulent shear stress transport between the inner and outer regions, is captured in the current results.

3) The surface normal penetration and stream-wise decay of isothermal jets of air, injected through oblique effusion holes, is well predicted. In addition, the prediction captures the coalescence of each jet with subsequent jets injected upstream.

4) Close to the combustion chamber surface, an entrainment of fluid from the mainstream flow towards the wall by the effusion jet is captured. Contra-rotating stream-wise vortices that form as a result of this entrainment are also well resolved.

5) The present results have been computed with what may be argued are affordable LES me-

shes (regarding a significant number of effusion hole rows, for example) in practical design calculations. This is encouraging for the extended use of large eddy simulation techniques to important industrial engineering applications.

References

- [1] Lefebvre A H. Gas turbine combustion [M]. 2nd ed. [S. l.]: Taylor & Francis, 1999.
- [2] Page G J, Zhao H, McGuirk J J. A parallel multi-block Reynolds-averaged Navier-Stokes method for propulsion installation applications [R]. Melbourne, Australia: 12th Int. Symp. on Air Breathing Engines 1, 2001:864-876.
- [3] Veloudis I, Yang Z, McGuirk J J, et al. Novel implementation and assessment of a digital filter based approach for the generation of LES inlet conditions [J]. J. Flow Turbulence and Combustion, 2007, 79: 1-24.
- [4] Smagorinsky J. General circulation experiments with the primitive equations, I: The basic experiment [J]. Monthly Weather Review, 1963, 91: 91-164.
- [5] Van Driest E R. On turbulent flow near a wall [J]. J. Aerospace Science, 1956, 23: 1007-1011.
- [6] Eriksson J G, Karlsson R I, Persson J. An experimental study of a two-dimensional turbulent wall jet [J]. Experiments in Fluids, 1998, 25: 50-60.
- [7] Karlsson R I, Eriksson J G, Persson J. An experimental study of a two-dimensional plane turbulent wall jet [R]. Report VU-S 93:B26, 1993.
- [8] Dejoan A, Leschziner M A. Large eddy simulation of a plane turbulent wall jet [J]. Physics of Fluids, 2005, 17 (2): 025102. 1-025102. 6.
- [9] Schlichting H. Boundary layer theory [M]. 6th ed. [S. l.]: McGraw Hill, 1968.
- [10] Miron P, Sabelnikov V, Most M, et al. Caractérisation aérodynamique d'un film de refroidissement généré par une plaque multiperforée [R]. Toulouse, France: Congress Francophone de Technique Laser, CFTL 2006, 19-22nd. 2006.
- [11] Mendez S, Eldridge J, Nicoud F. Numerical investigation and preliminary modeling of a turbulent flow over a multi-perforated plate [R]. In Center for Turbulence Research. Proceedings of the Summer Program, 2006.
- [12] Launder B E, Rodi W. The turbulent wall jet [J]. Progress in Aerospace Sciences, 1981, 19: 81-128.
- [13] Launder B E, Rodi W. The turbulent wall jet: measurements and modeling [J]. Annual Review of Fluid Mechanics, 1983, 15: 429-459.
- [14] George W L, Abrahamson H, Eriksson J G, et al. A similarity theory for the turbulent plane wall jet without external stream [J]. J. Fluid Mech., 2000, 425: 367-411.
- [15] Barenblatt G I, Chorin A J, Prostokishin V M. The turbulent wall jet: a triple-layered structure and incomplete similarity [J]. Proc. National Academy of Sciences of the United States of America, 2005, 102(25): 8850-8853.
- [16] Dejoan A, Wang C, Leschziner M A. Assessment of turbulence models for predicting the interaction region in a wall jet by reference to LES solution and budgets [J]. Flow, Turbulence and Combustion Journal, 2006, 77: 229-255.
- [17] Miron P. Étude expérimentale des lois de parois et du film de refroidissement produit par une zone multiperforée sur une paroi plane [D]. [S. l.]: Université de Pau et des Pays de l'Adour, 2005.
- [18] Yavuzkurt S, Moffat R J, Kays W M. Full-coverage film cooling - Part 1: Three-dimensional measurements of turbulence structure [J]. J. Fluid Mech., 1980, 101: 129-158.
- [19] Tyagi M, Acharya S. Large eddy simulation of film cooling flow from an inclined cylindrical jet [J]. Trans. ASME, J. Turbomachinery, 2003, 125: 734-742.
- [20] Dubief Y, Delcayre J. On coherent-vortex identification in turbulence [J]. J. Turbulence, 2000, 1: 1-22.
- [21] Luo K, Fan J, Cen K. Pressure-correlated dispersion of inertial particles in free shear flows [J]. Physics Review E, 2007, 75(4): 046309. 1-046309. 5.