Introduction

Film cooling is a process that is used to protect turbine vanes in a gas turbine engine from exposure to hot combustion gases. This tutorial illustrates how to set up and solve a film cooling problem using a non-conformal mesh. The system that is modeled consists of three parts: a duct, a hole array, and a plenum. The duct is modeled with a hexahedral mesh, and the plenum and hole regions are modeled using a tetrahedral mesh. These two meshes are merged together to form a "hybrid" mesh, with a non-conformal interface boundary between them.

Due to symmetry of the hole array, only a portion of the geometry is modeled in FLUENT, with symmetry applied to the outer boundaries. The duct contains a high-velocity fluid in streamwise flow (Figure 6.1). An array of holes intersects the duct at an inclined angle, and a cooler fluid is injected into the holes from a plenum. The coolant that moves through the holes acts to cool the surface of the duct, downstream of the injection. Both fluids are air, and the flow is classified as turbulent. The velocity and temperature of the streamwise and cross-flow fluids are known, and FLUENT is used to predict the flow and temperature fields that result from convective heat transfer.

In this tutorial you will learn how to:

- Merge hexahedral and tetrahedral meshes to form a hybrid mesh
- Create a non-conformal grid interface
- Model heat transfer across a non-conformal interface with specified temperature and velocity boundary conditions
- Calculate a solution using the segregated solver
- Plot temperature profiles on specified isosurfaces

Prerequisites

This tutorial assumes that you are familiar with the menu structure in FLUENT and that you have completed Tutorial 1 . Some steps in the setup and solution procedure will not be shown explicitly.

Problem Description

This problem considers a model of a 3D section of a film cooling test rig. A schematic is shown in Figures 6.1 and 6.2. The problem consists of a duct, 24.5 in long, with cross-sectional dimensions of 0.75 in \times 5 in. An array of uniformly spaced holes is located at the bottom of the duct. Each hole has a diameter of 0.5 inches, is inclined at 35 degrees, and is spaced 1.5 inches apart laterally. Cooler injected air enters the system through the plenum, with cross-sectional dimensions of 3.3 in \times 1.25 in.

Because of the symmetry of the geometry, only a portion of the domain needs to be modeled. The computational domain is shown in outline in Figure 6.2. The bulk temperature of the streamwise air (T_{∞}) is 273 K, and the velocity of the air stream is 20 m/s. The bottom wall of the duct that intersects the hole array is assumed to be a completely insulated (adiabatic) wall. The secondary (injected) air enters the plenum at a uniform velocity of 0.4559 m/s. The temperature of the injected air (T_{inject}) is 136.6 K. The properties of air that are used in the model are shown in Figure 6.2.

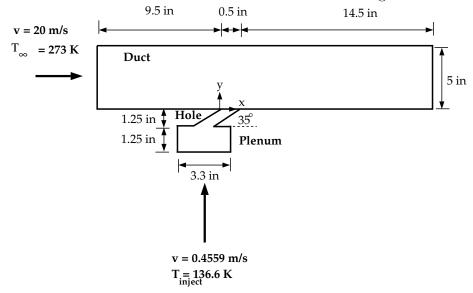


Figure 6.1: Schematic of the Problem, Front View

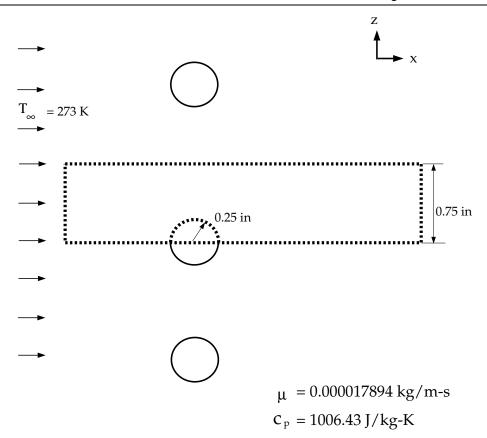


Figure 6.2: Schematic of the Problem, Top View

Setup and Solution

Preparation

- 1. Download non_conformal_mesh.zip from the Fluent Inc. User Services Center or copy it from the FLUENT documentation CD to your working directory (as described in Tutorial 1).
- 2. Unzip non_conformal_mesh.zip.

film_hex.msh and film_tet.msh can be found in the /non_conformal_mesh folder created after unzipping the file.

Step 1: Merging the Mesh Files

- 1. Start the 3D version of tmerge by typing utility tmerge -3d at the system prompt.
- 2. Provide the mesh file names film_tet.msh and film_hex.msh as prompted. Provide scaling of 1 and translations and rotations of zero for each mesh file. Save the new merged mesh file as filmcool.msh.

Append 3D grid files. tmerge3D Fluent Inc, Version 2.1.11			
Enter name of grid file (ENTER to conti	.nue) : [film_tet.msh]		
x,y,z scaling factor, eg. 1 1 1	: 1 1 1		
x,y,z translation, eg. 0 1 0	: 0 0 0		
rot axis (0,1,2), angle (deg), eg. 1 45	5 : 0 0		
Enter name of grid file (ENTER to conti	.nue) : film_hex.msh		
x,y,z scaling factor, eg. 1 1 1	: 1 1 1		
x,y,z translation, eg. 0 1 0	: 0 0 0		
rot axis (0,1,2), angle (deg), eg. 1 45	5 : 0 0		
Enter name of grid file (ENTER to conti	nue) : <enter></enter>		
Enter name of output file	: filmcool.msh		



The mesh files must be read into tmerge in this order for the tutorial to run as written. Otherwise, zone names and numbers will be assigned differently when the files are merged together. In general, however, you can specify files to be read into tmerge in any order.

Step 2: Grid

- 1. Start the 3D version of FLUENT.
- 2. Read in the mesh file filmcool.msh. File \longrightarrow Read \longrightarrow Case...
- 3. Check the grid.

Grid → Check

FLUENT will perform various checks on the mesh and will report the progress in the console window. Pay particular attention to the reported minimum volume. Make sure this is a positive number.

4. Scale the grid and change the unit of length to inches.

- Scale Grid		
Scale Factors	Unit Conversion	
X 0.0254	Grid Was Created In in	
Y 0.0254	Change Length Units	
Z 0.0254		
Domain Extents		
Xmin (in) _9,5	Xmax (in) 15	
Ymin (in) _2,5	Ymax (in) 5,000001	
Zmin (in) 👌	Zmax (in) 0,75	
Scale Un:	scale Close Help	

- (a) In the Unit Conversion drop-down list, select in to complete the phrase Grid Was Created In in (inches).
- (b) Click Scale to scale the grid.
- (c) Click Change Length Units to set inches as the working units for length. The final Domain Extents should appear as in the panel above.
- (d) Close the panel.

5. Display an outline of the 3D grid (Figure 6.3).

Display	\longrightarrow Grid
---------	------------------------

🗕 Grid Display 🏼 🗆		
Options Nodes Edges Faces Partitions 	Edge Type All Feature Outline	Surfaces
Surface Name Pa	Surface Types 📄 🚍 axis clip-surf exhaust-fan fan	
Outline Interior Display Colors Close Help		

- (a) In the Surfaces list, deselect symmetry-3, symmetry-5 and symmetry-tet.
- (b) Click Display.

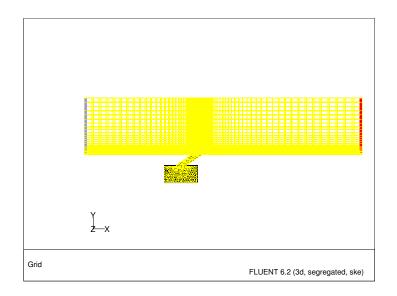


Figure 6.3: Hybrid Mesh for Film Cooling Problem

(c) Zoom in using your middle mouse button to get the view displayed in Figure 6.4.

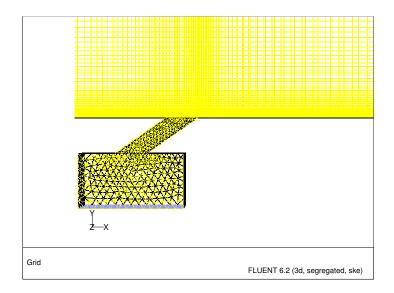


Figure 6.4: Hybrid Mesh (Zoomed-In View)

In Figure 6.4 you can see the quadrilateral faces of the hexahedral cells that are used to model the duct region, and the triangular faces of the tetrahedral cells that are used to model the plenum and hole regions, resulting in a hybrid mesh.

- Extra: You can use the right mouse button to check which zone number corresponds to each boundary. If you click the right mouse button on one of the boundaries in the graphics window, its zone number, name, and type will be printed in the FLUENT console window. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.
- 6. Close the Grid Display panel.

Step 3: Models

1. Keep the default solver settings.

 $\boxed{\text{Define}} \longrightarrow \boxed{\text{Models}} \longrightarrow \boxed{\text{Solver}}...$

- Solver		
Solver	Formulation	
Segregated	🛧 Implicit	
💠 Coupled	🕹 Explicit	
Space	Time	
l 🕹 20	🔷 Steady	
💠 Axisysseettic	🕹 Unsteady	
💠 Axisyawatric Sviri		
🗢 3D		
Velocity Formulation	1	
* Absolute		
💠 Relative		
Gradient Option	Porous Formulation	
◆ Cell-Based	🔷 Superficial Velocity	
✤ Node-Based	💠 Physical Velocity	
OK Cancel Help		

2. Enable heat transfer by activating the energy equation.

 $\fbox{Define} \longrightarrow \r{Models} \longrightarrow \r{Energy}...$

[]	Energy	
Energy		
J	Energy Equation	
	OK Cancel Help	
L		

3. Enable the standard k- ϵ turbulence model.

Viscous Model		
Model	Model Constants	
💠 Inviscid	Cmu	
💠 Laminar	0,09	
💠 Spalart-Allmaras (1 eqn)	C1-Epsilon	
🔷 k-epsilon (2 eqn)	1,44	
∻ k−omega (2 eqn)	C2-Epsilon	
	1,92	
✓ Detached Eddy Simulation	TKE Prandti Number	
	1	
k-epsilon Model	User-Defined Functions	
🔷 Standard	Turbulent Viscosity	
	none	
	Prandti Numbers	
Near-Wall Treatment	TKE Prandtl Number	
Standard Wall Functions	none 🔻	
Non-Equilibrium Wall Functions		
✤ Enhanced Wall Treatment		
Options		
☐ Viscous Heating		
OK Cancel Help		

- (a) Activate k-epsilon (2 eqn) under Model to expand the panel.
- (b) Click OK.

Step 4: Materials

Define \longrightarrow Materials...

1. Retain air as the Fluent Fluid Materials , and use the incompressible-ideal-gas law to compute density. Retain the default values for all other properties.

air flui Chemical Formula Flue air Mixi Door Properties	it Fluid Materials	Order Materials By Image: State of the state	
Chemical Formula Flue air Mixi Boom Properties Density (kg/m3) incompres Cp (j/kg-k) constant	it Fluid Materials	Chemical Formu Fluent Database.	
Properties Cp (j/kg-k) Constant	HT &	Fluent Database.	
Properties Density (kg/m3) incompres Cp (j/kg-k) constant		Fluent Database.	
Properties Properties Cp (j/kg-k) Constant		User-Defined Datab	se
Properties Density (kg/m3) incompres Cp (j/kg-k) constant	»	<u> </u>	
Density (kg/m3) incompres Cp (j/kg-k) constant			
Cp (j/kg-k) constant		TE	
Constant	sible-ideal-gas 👿 E&C		
Constant			
1006,43	constant		
Thermal Conductivity (w/m-k)	constant V Edit		
Lonstan			
	0.0242		
Viscosity (kg/m-s) constant	▼ E68	l P	
1,7894e-	5		
Change/Create Dele		Help	



Do not forget to click the Change/Create button after selecting incompressible-ideal-gas in the drop-down list for Density.

Step 5: Operating Conditions

1. Keep the default operating conditions.

 $\boxed{\text{Define}} \longrightarrow \text{Operating Conditions...}$

Operating Conditions	
Pressure	Gravity
Operating Pressure (pascal)	💷 Gravity
101325	
Reference Pressure Location	
X (in) ◊	
∀ (in) ◊	
Z (in)	
OK Cancel	Help

Step 6: Boundary Conditions

Define \longrightarrow Boundary Conditions...

1. Set the boundary conditions for the streamwise flow inlet (velocity-inlet-1).

	Velocity Inlet	
Zone Name		
velocity-inlet-duct		
Velocity Specification Metho	d Magnitude, Norma	I to Boundary
Reference Fram	e Absolute	V
Velocity Magnitude (m/s)	20	constant
Temperature (k)	273	constant 🛛 🔻
Turbulence Specification Method Intensity and Hydraulic Diameter		
Turbulence Intensity (%)		
Hydraulic Diameter (in)	5	
ОК	Cancel H	elp

- (a) Change the Zone Name from velocity-inlet-1 to velocity-inlet-duct.
- (b) Set the Velocity Magnitude to 20 m/s.
- (c) Set the Temperature to 273 K.
- (d) In the Turbulence Specification Method drop-down list, select Intensity and Hydraulic Diameter.
- (e) Set the Turbulence Intensity to 1% and the Hydraulic Diameter to 5 in.
- 2. Set the boundary conditions for the injected stream inlet (velocity-inlet-14).
 - (a) Change the Zone Name from velocity-inlet-14 to velocity-inlet-plenum.
 - (b) Set the Velocity Magnitude to 0.4559 m/s.
 - (c) Set the Temperature to 136.6 K.
 - (d) In the Turbulence Specification Method drop-down list, select Intensity and Viscosity Ratio.

- Velo	ocity Inlet	
Zone Name		
velocity-inlet-plenum		
Velocity Specification Method Ma	gnitude, Normal to Boundary	
Reference Frame Ab:	solute 🛛 🝸	
Velocity Magnitude (m/s) 0,4559	constant 🛛 🔻	
Temperature (k) 136,6	constant 🛛 🔻	
Turbulence Specification Method Intensity and Viscosity Ratio		
Turbulence Intensity (%) 1		
Turbulent Viscosity Ratio 10		
OK Cancel Help		

(e) Set the Turbulence Intensity to 1% and keep the Turbulent Viscosity Ratio default of 10.

In the absence of any identifiable length scale for turbulence, the Intensity and Viscosity Ratio method should be used.

See Chapter 11 of the User's Guide for more information on how to set the boundary conditions for turbulence.

3. Set the boundary conditions for the flow exit (pressure-outlet-1).

- Pressure Outlet			
Zone Name			
pressure-outlet-duct			
Gauge Pressure (pascal) ◊ constant 🕎			
Radial Equilibrium Pressure Distribution			
Backflow Total Temperature (k) 273 constant			
Backflow Direction Specification Method Normal to Boundary			
Turbulence Specification Method Intensity and Viscosity Ratio			
Backflow Turbulence Intensity (%)			
Backflow Turbulent Viscosity Ratio			
Target mass-flow rate			
OK Cancel Help			

- (a) Change the Zone Name from pressure-outlet-1 to pressure-outlet-duct.
- (b) Keep the default setting of $0\ {\rm Pa}$ for ${\sf Gauge\ Pressure}.$
- (c) Set the Backflow Total Temperature to 273 K.
- (d) In the Turbulence Specification Method $\operatorname{drop-down}$ list, select Intensity and Viscosity Ratio.
- (e) Set the Backflow Turbulence Intensity to 1% and keep the Backflow Turbulent Viscosity Ratio default of 10.
- 4. Set the conditions for the fluid in the duct (fluid-9).

Fluid		
Zone Name		
fluid-duct		
Material Name air 🛒 Edit		
Source Terms		
☐ Fixed Values		
Porous Zone		
🗖 Laminar Zone		
Motion Source Versus Fixed Values Poroas Zone Reaction		
Rotation-Axis Origin Rotation-Axis Direction		
X (in) 10 X 0		
∀ (in) ○ ∀ ○		
Z (in) • Z 1		
Motion Type Stationary		
OK Cancel Help		

- (a) Change the Zone Name from fluid-9 to fluid-duct.
- (b) Keep the default selection of air as the Material Name.

- 5. Set the conditions for the fluid in the plenum and hole (fluid-17).
 - (a) Change the Zone Name from fluid-17 to fluid-plenum.
 - (b) Keep the default selection of air as the Material Name.

-		Flui	id		
Zone Name					
fluid-plenum	ì				
Material Name	air		Fdit		
Source Terms					
🔲 Fixed Value	☐ Fixed Values				
🗆 Porous Zone	💷 Porous Zone				
🗆 Laminar Zon	💷 Laminar Zone				
Motion See	ve Verbiy	Fixed Vah	ees Poroe	39 Zoste 🗍	Reaction
Rotation-Ax	is Origin	Rol	tation-Axis	Direction	
X (in) 🔍		×	•		
Y (in) 🔍		v	• •		
Z (in) 🔍		z	1		
Motion Type Stationary					
	OK Cancel Help				

6. Keep the default boundary conditions for the plenum and hole wall (wall-15).

-	Wall	
Zone Name		
wall-15		
Adjacent Cell Z	ne	
fluid-plenum		
Thermal DP	Momentum Species Hadiation HDS Grandar	
Thermal Condi	ons	
🔶 Heat Flux	Heat Flux (w/m2) 0	constant 🛛 🔻
💠 Temperat	re Wall Thickness (in) 🛇	
💠 Convectio		
	Heat Generation Rate (w/m3)	constant 🗾 🝸
💠 Mixed		Shell Conduction
Material Name	V Edit	
	OK Cancel Help	

7. Define the zones on the non-conformal boundary as interface zones.

The non-conformal grid interface contains two boundary zones: wall-1 and wall-12. wall-1 is the bottom surface of the duct, and wall-12 represents the hole through which the cool air is injected from the plenum (Figure 6.5). These boundaries were defined as walls in the original mesh files, film_hex.msh and film_tet.msh, and must be redefined as interface boundary types.

- (a) Open the Grid Display panel.
- (b) Select wall-1 and wall-12 under Surfaces.
- (c) Click Display.
 - i. Activate bottom view.

Display \longrightarrow Views...

A. Select bottom under Views and click Restore.

Zoom in using middle mouse button to get the view as shown below.

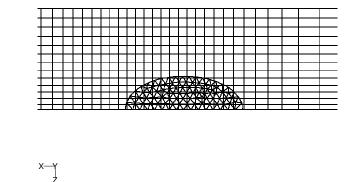
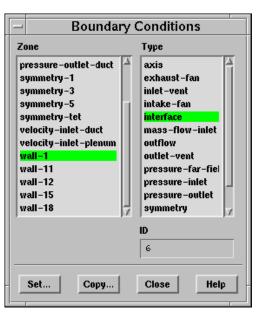




Figure 6.5: Grid for the wall-1 and wall-12 Boundaries

- 8. Change the Type for wall-1 and wall-12 to interface.
 - (a) Select wall-1 in the Zone list and choose interface as the new Type.



- (b) Click Yes when asked OK to change wall-1's type from wall to interface?.
- (c) Change the Zone Name to interface-duct.

-	interface 🏼 🗖	I
z	one Name	
	interface-duct	
<u> </u>		-
	OK Cancel Help	

- (d) Repeat this procedure to convert wall-12 to an interface boundary zone named interface-hole.
- 9. Close the Boundary Conditions panel.

Step 7: Grid Interfaces

In this step, you will create a non-conformal grid interface between the hexahedral and tetrahedral meshes.

Grid Interfaces Grid Interface Interface Zone 1 Interface Zone 2 interface-hole interface-duct junction nction interface-duct interface-ducl interface-hole interface-hole Interface Wall Zone 1 Interface Type Boundary Zone 1 wall-11 💷 Periodic **Boundary Zone 2** Interface Wall Zone 2 Coupled wall-18 List Close Create Delete Help

Define \longrightarrow Grid Interfaces...

1. Select interface-hole in the Interface Zone 1 list.



When one interface zone is smaller than the other, it is recommended that you choose the smaller zone as Interface Zone 1.

- 2. Select interface-duct in the Interface Zone 2 list.
- 3. Enter the name junction under Grid Interface.
- 4. Click Create.
 - Note: In the process of creating the grid interface, FLUENT creates two new wall boundary zones: wall-11 and wall-18. You will not be able to display these walls.

wall-11 is the non-overlapping region of the interface-hole zone that results from the intersection of the interface-hole and interface-duct boundary zones, and is listed under Boundary Zone 1 in the Grid Interfaces panel. wall-11 is empty, since interface-hole is completely contained within the interface-duct boundary.

wall-18 is the non-overlapping region of the interface-duct zone that results from the intersection of the two interface zones, and is listed under Boundary Zone 2 in the Grid Interfaces panel.



In general, you will need to set boundary conditions for wall-18 (since it is not empty). In this case, default settings are used.

Step 8: Solution

1. Set the solution parameters.

- Solution Controls			
Equations 📃 📃	Under-Relaxation Factors		
Flow Turbulence	Pressure	0,3	
Energy	Density	1	
	Body Forces	1	
Pressure-Velocity Coupling	Momentum	0.7	
	Discretization		
SIMPLE	Pressure	Standard	
	Momentum	Second Order Upwind	
	Turbulence Kinetic Energy	Second Order Upwind	
	Turbulence Dissipation Rate	Second Order Upwind	
ок	Default Cancel	Help	

- (a) Under Discretization, select Second Order Upwind for Momentum and Turbulence Kinetic Energy.
- (b) Scroll down the list and select Second Order Upwind for Turbulence Dissipation Rate and Energy.
- 2. Enable the plotting of residuals.

Solve \longrightarrow Monitors \longrightarrow Residual...

- (a) Under Options, select Plot.
- (b) Click the OK button.

Residual Monitors				
Options	Storage		Plotting	
F Print	Iterations	1000	Window	w 💿 🌲
	Normalizatio	on	Iterations	1000
	🗆 Normali	ize 🌹 Scale	Axes	Curves
Residual		Check C Convergence C	Convergence Criterion	Î
continuit	ty 📕	– [0,001	
x-velocit	y 📕	– [0,001	
y-velocit	:y 📕	– [0,001	
z-velocit	y 📕	– [<u>1</u> 0,001	ľ
energy		– [1e-06	,
ОК	Plot	Renorm	Cancel	Help

3. Initialize the solution.

 $\fbox{Solve} \longrightarrow \fbox{Initialize} \longrightarrow \fbox{Initialize}...$

- Solution Initia	alization
Compute From	Reference Frame
V	✤ Relative to Cell Zone
	💠 Absolute
Initial Values	
Gauge Pressure (pascal)	<u>ہ</u>
X Velocity (m/s)	20,00001
¥ Velocity (m∕s)	0
Z Velocity (m/s)	0
	F
Init Reset Apply	Close Help

- (a) Select velocity-inlet-duct in the Compute From drop-down list.
- (b) Click Init, and Close the panel.

4. Save the case file (filmcool.cas).

 $\mathsf{File} \longrightarrow \mathsf{Write} \longrightarrow \mathsf{Case...}$

5. Start the calculation by requesting 250 iterations. Solve \longrightarrow Iterate...

-	lterate
It	eration
	Number of Iterations 250
	Reporting Interval 🚺 📥
	UDF Profile Update Interval 🚺 📥
	iterate Apply Close Help -

- (a) Set the Number of Iterations to 250.
- (b) Click Iterate.
- **Note:** During the first few iterations, the console window will report that turbulent viscosity is limited in a couple of cells. This message should go away as the solution converges and the turbulent viscosity approaches more reasonable levels.

The solution will converge after about 135 iterations.

6. Save the case and data files (filmcool.cas and filmcool.dat).

 $[\mathsf{File}] \longrightarrow \mathsf{Write} \longrightarrow \mathsf{Case} \And \mathsf{Data...}$

Note: If you choose a file name that already exists in the current directory, FLU-ENT will prompt you for confirmation to overwrite the file.

Step 9: Postprocessing

1. Display filled contours of static pressure (Figure 6.6).

 $\mathsf{Display} \longrightarrow \mathsf{Contours}...$

-	Contours
Options	Contours of
Filled	Pressure
📕 Node Values	Static Pressure
📕 Global Range	Min Max
📕 Auto Range	
🔲 Clip te Resige	Surfaces = =
Draw Profiles	
Draw Grid	velocity-inlet-duct velocity-inlet-plenum wall-11
Levels Setup	wall-15
20 🛔 1	wall-18
Surface Name Pattern	Surface Types 📃 📃
	axis
	clip-surf exhaust-fan
Match	fan
Display Com	pute Close Help

- (a) Select Filled under Options.
- (b) Select Pressure... and Static Pressure in the Contours of drop-down lists.
- (c) In the Surfaces list, select interface-duct and interface-hole.
- (d) Scroll down the Surfaces list and select symmetry-1, symmetry-tet, and wall-15.

(e) Reset the view to the default view.

 $\mathsf{Display} \longrightarrow \mathsf{Views...}$

i. Click Default under Actions.

Views		
Views	Actions	Mirror Planes 🔳 🚍
back bottom	Default	symmetry – 3 symmetry – 1
front isometric	Auto Scale	symmetry –5
left	Save	symmetry -tet
right top	Restore	Define Plane
	Delete	Periodic Repeats
Save Name	Read	Consect majores
view-0	Write	Define
1 0160 0	Write	
Apply	Camera CI	ose Help

- ii. Close the panel.
- (f) In the Contours panel, click Display.
- (g) Zoom in on the view to get the display shown in Figure 6.6.

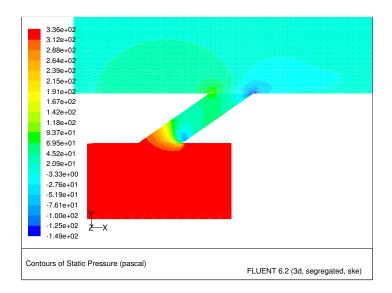


Figure 6.6: Contours of Static Pressure

Note the high/low pressure zones on the upstream/downstream sides of the coolant hole, where the jet first penetrates the primary flow in the duct.

2. Display filled contours of static temperature (Figures 6.7 and 6.8).

Display	\longrightarrow Contours
---------	----------------------------

-	Contours	
Options	Contours of	
Filled	Temperature	V
📕 Node Values	Static Temperatu	re 🔻
📕 Global Range	Min	Max
🔲 Auto Range	0	273,096
☐ Clip to Range	Surfaces	 ∎ =
Draw Profiles	velocity-inlet-duct	
Draw Grid	velocity-inlet-pl wall-11	enum
Levels Setup	wali-15 wali-18	
20 🗣 1	Surface Tunes	
Surface Name Pattern	axis	
	clip-surf exhaust-fan	
Match	fan	¥
]	
Display Com	pute Close	Help

- (a) Select Temperature... and Static Temperature in the Contours of drop-down lists.
- (b) Under **Options**, deselect **Auto Range** so that you can change the maximum and minimum temperature gradient values to be plotted.
- (c) Keep the default Min value of 0.
- (d) Enter a new Max value of 273.096.
- (e) Under Options, deselect Clip to Range.
- (f) Click Display.
- (g) Zoom in on the view to get the display shown in Figure 6.8.

Figures 6.7 and 6.8 clearly show how the coolant flow insulates the bottom of the duct from the higher-temperature primary flow.

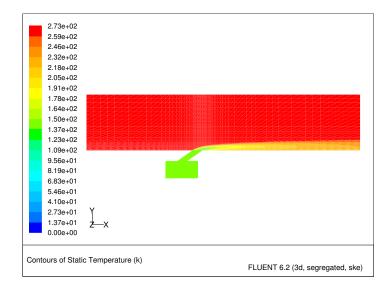


Figure 6.7: Contours of Static Temperature

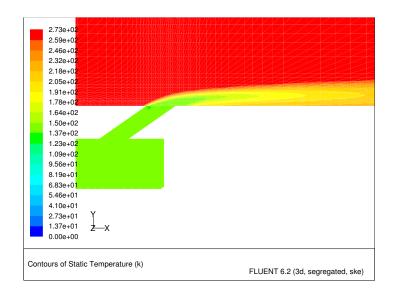


Figure 6.8: Contours of Static Temperature (Zoomed-In View)

3. Display the velocity vectors (Figure 6.9).

Display	\longrightarrow Vectors
---------	---------------------------

- Vectors		
Options	Vectors of	
□ Nede Values	Velocity 🔻	
📕 Global Range	Color by	
📕 Auto Range	Velocity	
💷 Chip te Renge	Velocity Magnitude	
📕 Auto Scale		
💷 Draw Grid		
Style arrow	Surfaces = =	
	symmetry-tet	
Scale 2	velocity-inlet-duct velocity-inlet-plenum	
Skip 🔍 📥	wall-11	
Vector Options	wali-15 wali-18	
Custom Vectors	Surface Types 📑 🗐	
Surface Name Pattern	axis	
	clip-surf exhaust-fan	
	fan y	
Match		
Display Compu	te Close Help	

- (a) Select Velocity... and Velocity Magnitude in the Color by drop-down lists.
- (b) Change the Scale to 2.

This will enlarge the vectors that are displayed, making it easier to view the flow patterns.

- (c) In the Surfaces list, select interface-duct and interface-hole.
- (d) Scroll down the Surfaces list and select symmetry-1, symmetry-tet, and wall-15.
- (e) In the Vectors panel, click Display.
- (f) Zoom in on the view to get the display shown in Figure 6.9.

The flow pattern in the vicinity of the coolant hole shows the level of penetration of the coolant jet into the main flow. Notice that the velocity field varies smoothly across the non-conformal interface.

	2.13e+01	
	2.02e+01	
	1.92e+01	
	1.81e+01	
	1.71e+01	
	1.60e+01	
	1.49e+01	
	1.39e+01	
	1.28e+01	
	1.17e+01	
	1.07e+01	
	9.60e+00	
	8.53e+00	
	7.47e+00	
	6.40e+00	
	5.33e+00	
	4.27e+00	
	3.20e+00	
	2.14e+00 Y	
	1.07e+00 <u>z x</u>	
	9.16e-03	
	and the second	
Velo	ocity Vectors Colored By Velocity Magnitude (m/s)	
. 510	, · · · · · · · · · · · · · · · · ·	FLUENT 6.2 (3d, segregated, ske)

Figure 6.9: Velocity Vectors

- 4. Plot the temperature profile along a horizontal cross-section of the duct, 0.1 inches above the bottom.
 - (a) Create an isosurface on the duct surface at y = 0.1 in.

Surface \longrightarrow Iso-Surface			
	- Iso-Surface		
Surface of Constant	From Surface 📑 🗐		
Grid	▼ int_interior-1		
Y-Coordinate	▼ interface - hole		
Min Max	interior-16 pressure-outlet-duct		
• •	symmetry-1		
Iso-Values	From Zones		
0.1	fluid-duct fluid-plenum		
	_		
New Surface Name			
y=0.1in			
Create Compute	Manage Close Help		

- i. Select Grid... and Y-Coordinate in the Surface of Constant drop-down lists.
- ii. Enter y=0.1in under New Surface Name.

- iii. Enter $0.1~{\rm for}$ Iso-Values.
- iv. Click Create.
- (b) Create an XY plot of static temperature on the isosurface.

 $Plot \longrightarrow XY Plot...$

- i. Keep the default Plot Direction.
- ii. Select Temperature... and Static Temperature in the Y-Axis Function drop-down lists.
- iii. Scroll down the $\mathsf{Surfaces}$ list and select $y{=}0.1in.$

Solution XY Plot			
Options	Plot Direction	Y Axis Function	
F Node Values	X 1	Temperature	
Position on X Axis	v •	Static Temperature	
Position on Y Axis	z	X Axis Function	
■ Write to File		Direction Vector	
Order Poisto		Surfaces	
File Data 🔳 🗐			
File Data 📃 📃		symmetry - tet velocity - inlet - duct	
		velocity-inlet-plenum	
		wali-11 wali-15	
		wall-18	
	Load File	y=0.1in	
	Lotarnem		
	Free Data		
Plot Axes Curves Close Help			

iv. Click Plot.

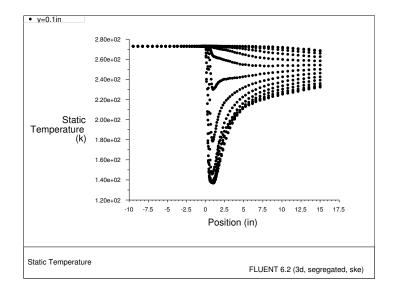


Figure 6.10: Static Temperature at y=0.1 in

In this plot you can see how the temperature of the fluid changes as the cool air from the injection hole mixes with the primary flow. As expected, the temperature is coolest just downstream of the hole. Note that you could also make a similar plot on the lower wall itself, to examine the wall surface temperature.

Summary

This tutorial demonstrates how FLUENT's non-conformal grid interface capability can be used to handle hybrid meshes for complex geometries, such as the film cooling hole configuration examined here. One of the principal advantages of this approach is that it allows you to merge existing component meshes together to create a larger, more complex mesh system, without requiring that the different components have the same node locations on their shared boundaries. Thus, you can perform parametric studies by merging the desired meshes, creating the non-conformal interface(s), and solving the model. For example, in the present case, you can

- Use a different hole/plenum mesh
- Reposition the existing hole/plenum mesh
- Add additional hole/plenum meshes to create aligned or staggered multiple hole arrays