

Multiphysics Computational Analysis of a Perforated Plate Cooling Flow

Justin M. Pesich^{*}, Nicholas J. Georgiadis[†] and Mark P. Wernet[‡]
NASA Glenn Research Center, Cleveland, OH, 44135

A multiphysics analysis of a film cooling flow was conducted using a computational fluid dynamics (CFD) solver in conjunction with a conjugate heat transfer (CHT) solver. The film cooling problem consisted of a square nozzle producing a heated subsonic jet over a perforated plate with three sections of 45 cooling holes each. Results from the multiphysics analysis were compared to a CFD-only approach and experimental results. Reynolds-averaged Navier-Stokes (RANS) computations were employed first, with and without the CHT solver. The multiphysics approach provided much closer agreement with experimentally measured plate temperatures than the CFD-only approach. Comparisons of computed velocities and temperatures were also made with experimental data. Hybrid RANS-large eddy simulation calculations employing the Delayed Detached Eddy Simulation (DDES) approach were also conducted within the multiphysics framework. The DDES approach was unable to resolve turbulent structures in the near wall regions. Hence, the RANS part of the DDES model was used in near wall regions, and produced surface temperatures along with velocity and temperature profiles close to the wall that were very similar to the RANS results. The DDES approach did resolve structures in the outer portion of the boundary layer especially near the end of the plate where the boundary layer was thicker. The key result was the improved surface temperatures produced by the multiphysics approach employing CHT over the fluid-only approach.

Nomenclature

Acronyms

BCD	Bounded-Central Differencing
BR	Blowing Ratio based on total mass flow rates and areas = $(m_{inj}/m_j) \cdot (A_j/A_{inj}) \sim (\rho U)_{inj}/(\rho U)_j$
CFD	Computational Fluid Dynamics
CFL	Courant-Friedrichs-Lewy
CHT	Conjugate Heat Transfer
DDES	Delayed Detached Eddy Simulation
LES	Large Eddy Simulation
NPR	Nozzle Pressure Ratio: p_{0j}/p_∞
PIV	Particle Image Velocimetry
RANS	Reynolds-averaged Navier Stokes
rms	Root Mean Square
SHJAR	Small Hot Jet Acoustic Rig
THX	Turbulent Heat Flux
TR	Temperature ratio: T_j/T_∞

Symbols

A_{inj} total injector tube area (135 holes) = 109.4 mm²

^{*} Thermal and Fluids Engineer, Fluid and Cryogenic Systems Branch.

[†] Aerospace Engineer, Inlets and Nozzles Branch, Associate Fellow.

[‡] Senior Research Engineer, Optics and Photonics Branch, Associate Fellow.

A_j	nozzle area at jet exit = 4634 mm ²
D	injector hole diameter = 1.016 mm
L_{plate}	plate length = 304.8 mm
m_{inj}	mass flow rate of the air supplied through the cooling holes [kg/s]
m_j	mass flow rate of the heated air from the square nozzle [kg/s]
M	Mach number
M_j	jet exit Mach number
p_{0j}	jet total pressure [Pa]
p_∞	ambient static pressure [Pa]
Pr_t	turbulent Prandtl number
T	static temperature
T'	rms temperature [K]
T_j	temperature at jet exit [K]
T_∞	ambient static temperature [K]
t	time [s]
u	axial velocity component [m/s]
u'	rms of axial velocity component [m/s]
U_{inj}	area-averaged velocity magnitude of the injected cooling flow [m/s]
U_j	axial velocity component of jet at nozzle exit [m/s]
v	vertical velocity component [m/s]
w	transverse velocity component [m/s]
x	axial distance from nozzle exit
y	vertical distance from plate surface
y^+	nondimensional wall distance
z	transverse coordinate from square nozzle centerline
ΔT	difference between jet exit temperature and ambient temperature = $(T_j - T_\infty)$ [K]
ρ_{inj}	gas density of the injected cooling flow [kg/m ³]
ρ_j	gas density of jet at nozzle jet exit [kg/m ³]
τ	nondimensional time

I. Introduction

Research into improving turbulence modeling for computational fluid dynamics (CFD) codes has been a focus of aeronautics research programs for many years, and is currently being sponsored by NASA's Transformative Tools and Technologies project. However, work into turbulence modeling for problems where turbulent heat flux is important has been rather limited. This is primarily due to the difficulty of obtaining detailed time-resolved measurements of temperature where there is turbulent transport of heat. As discussed in Refs. [1-3], standard Reynolds-averaged Navier-Stokes (RANS) based techniques perform poorly when considering the turbulent heat flux and temperatures in propulsion flows involving turbulent shear layers.

More accurate computational modeling techniques would greatly improve the ability to design effective propulsion flow components that require active cooling. The most common form of cooling is known as film cooling, which is most frequently used for turbine blades, combustion chamber walls, and the liners in afterburning engines. A review paper by Bogard and Thole [4] discusses the fundamentals of film cooling. The standard practice for RANS analyses of heated propulsion flow components is to use standard two-equation modeling and then employ relatively simple boundary conditions for wall surface temperatures or heat flux. In recent years, conjugate heat transfer (CHT) analyses have shown improvements over the standard RANS method and simple boundary condition technique [5]. In this paper, a fully-coupled CFD and thermal analysis of a benchmark cooling flow experiment was performed and compared with standard CFD-only analyses using Siemens Simcenter STAR-CCM+ commercial CFD code. In addition, a hybrid RANS-Large Eddy Simulation (LES) using the delayed detached eddy simulation (DDES) technique was utilized within the CFD-CHT model framework.

First, background of the recent film cooling experiment that is evaluated in this paper is presented. Next, a description of the computational model is provided. Finally, results of the CFD-only and thermally coupled CFD-CHT analyses are compared to the experimental results. Comparisons of plate surface temperature and streamwise centerline flow profiles of velocity and temperature for both mean and root-mean-squared (rms) quantities are provided and discussed.

II. Experimental Background

A series of tests have been underway during the past few years at NASA Glenn Research Center to collect data that can improve the knowledge base for high temperature turbulent flows in propulsion systems, and ultimately improve computational modeling of such flows. The tests are referred to as the Turbulent Heat Flux (THX) experiments, of which there are five phases. In the first phase (2015), single and multiple cooling hole configurations were tested in the NASA Glenn Engine Research Building SW6 test cell [6]. Phases 2-5 have utilized the Small Hot Jet Acoustic Rig (SHJAR) in the Aeroacoustic Propulsion Laboratory. Phase 2 (2016) examined a series of convergent round nozzle cases using a new spontaneous rotational Raman scattering spectroscopy technique to obtain mean and rms temperature measurements of the turbulent jet flow fields. The subsonic jet flows had been previously tested in the SHJAR to collect turbulent velocity statistics using Particle Image Velocimetry (PIV) [7]. This PIV data is already being used extensively by the CFD and aeroacoustics communities for turbulence modeling validation and modeling improvement efforts. The new Raman temperature measurements are discussed in Ref. [8].

In the third phase of the THX experiments [9], a square nozzle delivering a hot jet exhaust over a rectangular plate equipped with a single cooling hole providing a cooling film on the plate surface was tested where PIV and the Raman spectroscopy techniques were used to acquire velocity and temperature measurements, respectively. In the fourth phase of the THX experiments, the same square nozzle was used with a plate having three patches of holes injecting unheated air to provide film cooling to the plate surface. It is this experimental configuration that was analyzed in the computational effort described in this paper. Phase 5 examined heated supersonic convergent-divergent round jets.

The phase 4 experiment was conducted in the NASA Glenn SHJAR, where the test article is shown in Fig. 1, and is described in more detail in Ref. [10]. An exhaust nozzle transitioned from a circular inlet to a square outlet through a series of super ellipsoids. Hot exhaust blew over the plate with three patches containing 45 cooling holes each. This configuration was designed to replicate a realistic nozzle or turbine configuration, for which there would be a large number of similar sized cooling holes that would create a cooling film on the surface and protect the material from the hot jet exhaust. Extensive data were obtained for a series of test cases with variations in the nozzle exhaust temperature and Mach number along with overall cooling air blowing ratio (BR). Both PIV and Raman measurements were used to acquire high quality, spatially-resolved measurements of the mean and rms velocities as well as the mean and rms temperatures in the film cooling flow field. In addition to off-body flow field measurements, infrared thermography (IR) and thermocouple measurements on the plate surface enabled estimates of the film effectiveness. All of the phase 4 experimental results are presented in Ref. [10].

A total of three different nozzle flow conditions, referred to as Set Points, were tested across a range of blowing ratios, of which only Set Point 23 is investigated here. The cases examined in this computational effort all used the same hot nozzle operating conditions listed in Table 1, but varied the cooling, characterized by the blowing ratio. This $BR = (m_{inj}/m_j) \cdot (A_j/A_{inj}) \sim (\rho_{inj} U_{inj})/(\rho_j U_j)$ was based on total mass flow rates and areas for 145 injector holes. A view looking down on the plate with the three patches of cooling holes is shown in Fig. 2. The red circles indicate the thermocouple locations embedded in the top surface of the plate. The empty blue circles indicate the locations of the thermocouples mounted on the underside of the plate. The vertical green lines show the planes where cross stream stereo-PIV data were collected. The horizontal green line through the center of the plate indicates where streamwise PIV data were collected. Raman temperature surveys were performed at the intersections of the solid green lines and the centerline streamwise PIV plane.

Table 1. Operating conditions

Set Point	TR	NPR	M_j	BR	Nozzle mass flow rate (kg/s)	Injector mass flow rate (kg/s)
23	1.765	1.103	0.376	0.0	0.53	0.0
23	1.765	1.103	0.376	1.0	0.53	0.0124
23	1.765	1.103	0.376	2.0	0.53	0.0249

III. Computational Model

The computational domain contains the test article enclosed inside a cylindrical body illustrated in Fig. 3. The cylindrical body was shaped with a diverging taper to mitigate flow exiting the domain on the rounded surface. The nozzle was modeled starting at the junction with the facility supply pipe. The circular nozzle inlet is coplanar with one end of the cylindrical enclosure. The other end of the enclosure is extended approximately 10 plate lengths

downstream. The enclosure boundaries were placed at a sufficient distance away from the nozzle/plate assembly such that the far-field boundary conditions would have no influence on the flow near the test article. The upstream and rounded surfaces of the cylinder are assigned freestream conditions at ambient pressure and temperature, and the downstream surface is set to a pressure outlet at ambient pressure and temperature in case of backflow. A freestream Mach number of 0.02 was set to prevent any convergence and/or stability issues. Previous simulations of jets exhausting into quiescent ambient [11] indicated that a freestream flow of this speed would yield jet decay very similar to a freestream with no flow ($Mach = 0$). The nozzle inflow condition was set to the stagnation pressure and temperature measured in the experimental hardware. For the injector flow, the stagnation pressure set at the inflow was iterated upon during a particular blowing ratio calculation, until the plenum mass flow rate from the experiment for that blowing ratio was matched to within 10 percent. In addition, the plenum inlet stagnation temperature was set to the experimentally measured value.

Two different modeling approaches were considered: (1) only the fluid domain was modeled using adiabatic wall boundary conditions, and (2) one fluid region and one solid region with conformal interfaces between boundaries to account for conjugate heat transfer between each physical domain. For the multiphysics simulations employing CHT, the energy equation is solved in both domains by balancing the heat transfer at each fluid-solid interface. For each approach, a family of polyhedral grids with several refinement levels were created, where the size of each grid and the blowing ratios considered are listed in Table 2. In STAR-CCM+, the polyhedral cell topology is the only grid type that can automatically create conformal interfaces between domains. The fluid-solid fine grid required some optimization in preparation for the DDES case which coincidentally closely matched the fluid-only fine grid cell count. An extra-fine (XFine) fluid-solid grid was created for an additional DDES case for comparison. Grids were packed to solid surfaces such that the y^+ corresponding to the first point off the wall did not exceed one. A comparison of the fluid-only coarse and fluid-solid fine grids is shown in Fig. 4.

Sutherland's Law was employed for the gas dynamic viscosity and thermal conductivity. The ratio of specific heats was set to 1.4 and the turbulent Prandtl number, Pr_t , was set to 0.9, which is the default in most RANS solvers. The solid region is modeled as Stainless Steel 316 with a constant density of 7900 kg/m^3 and constant specific heat of 495 J/kg-K . The temperature-dependent thermal conductivity was incorporated as a lookup table in STAR-CCM+ using values from the Aerospace Structural Metals Handbook Code 1307 [12]. Note that radiation heat transfer was neglected for both modeling approaches.

Table 2. Computational grids and blowing ratios considered

Grid	Fluid-Only # cells	Fluid-Solid # cells	Blowing Ratio
Coarse	26.3 million	29.9 million	0.0, 1.0, 2.0
Medium	70.3 million	78.9 million	1.0
Fine	143.3 million	143.3 million	1.0
XFine	N/A	321.4 million	1.0

IV. Results and Discussion

A. Baseline RANS CFD Analysis

CFD simulations of the experimental configuration were run using Siemens Simcenter STAR-CCM+ Version 15.02.007-R8 [13] commercial CFD code. The simulations were run in Reynolds-averaged Navier-Stokes (RANS) mode with the $k-\omega$ Menter shear stress transport (SST) model [14, 15] using the second order upwind discretization scheme and coupled implicit solver. The coupled implicit solver uses a pseudo-time-marching approach where a constant Courant-Friedrichs-Lewy (CFL) number or an automatic CFL number control can be used to march towards the steady-state solution. The automatic CFL control, which was used in this study, varies the CFL number between user-specified maximum and minimum values to promote fast convergence while maintaining stability. Each grid and blowing ratio combination was run to convergence where several quantities were monitored such as plate surface temperature, injector flow rate, and maximum velocity. Notably, the plate surface temperature converged the slowest and the simulation was stopped once the maximum temperature fluctuation was less than 0.005 Kelvin over 100 iterations at four computational probe locations on the plate.

Nozzle and plenum boundary conditions applied for each blowing ratio with comparisons of measured and calculated flow rates are presented in Table 3. Note that the experimental nozzle flow rate was matched to within 2 percent, and the maximum error for the injector flow rate was 8 percent, and in most cases was significantly lower than this maximum.

Table 3. Nozzle and plenum boundary conditions with comparisons of flow rates

Variable	BR0	BR1	BR2
Ambient Temp, K	296		
Ambient Pressure, kPa	98.960		
Nozzle Temp, K	537		
Nozzle Pressure, kPa	109.130	109.130	109.130
Plenum Temp, K	N/A	304	299
Plenum Pressure, kPa	N/A	117.185	161.744
EXP m_{inj} , kg/s	0.0	0.0124	0.0249
Fluid-Solid m_{inj} , kg/s	0.0	0.0131	0.0262
Fluid-Only m_{inj} , kg/s	0.0	0.0134	0.0266

Grid independence studies were carried out for the fluid-only and fluid-solid grid families. The streamwise centerline axial velocity and temperature flow profiles at the most aft measurement location of $x=270.3$ mm for $BR=1$ are displayed in Fig. 5. The full nozzle flow profile is provided for velocity while the plots of temperature only show the boundary layer profile. All three grids for both modeling approaches are shown to be in good agreement, with the coarse grid displaying minor discrepancies compared to the medium and fine levels. This was true for all measurement locations. Note that the grid independence study was carried out only for the $BR=1$ case as indicated in Table 2. All subsequent RANS results presented herein are from the coarse grids for $BR=0$ and $BR=2$, and fine grids for $BR=1$.

Comparisons of the centerline surface temperature and plate surface temperature contours for $BR=0$, 1, and 2 are presented in Fig. 6, Fig. 7, and Fig. 8, respectively. The thermally coupled CFD-CHT results compare reasonably well to the experiment and significantly improved the prediction over the CFD-only adiabatic wall boundary condition results. For all three blowing ratios, the fluid-solid prediction captures the centerline surface temperature trends along the plate with the exception of the very aft end. The experimental IR data show an increase in temperature, particularly aft of $x=270.3$ mm, while the fluid-solid CFD results show a decrease in temperature. There are several potential explanations for this behavior: (1) the jet core mixes with the film cooling flow downstream of the last cooling patch and the RANS simulations underpredicts this behavior, (2) the plate structure begins to thin out at $x=288.2$ mm (see Fig. 1) resulting in less thermal mass to absorb heat and raises the temperature, and (3) the last thermocouple used to calibrate the IR measurements was at $x=270.3$ mm, so any IR data aft of that location is extrapolated. Regarding (2), the CHT simulation should be able to predict the temperature increase, however the applied $Mach = 0.02$ freestream flow in the CFD simulation may be cooling the underside of the plate reducing this effect. It should be noted that the biggest discrepancies between the thermocouple measurements and IR data were on the plate overhang ($\sim 15K$), and that Set Point 23 was the first data set obtained. Initially the IR camera focus was not optimized which was corrected for later data sets [10]. Overall, the CFD-CHT results show that a multiphysics simulation framework is required to accurately predict material temperatures.

Streamwise centerline flow profiles for $BR=0$ and $BR=2$ comparing the two modeling approaches and experimental measurements are shown in Fig. 9 – Fig. 12. Nondimensional velocity and temperature profiles at four locations along the plate ($x=76.8$ mm, $x=138.4$ mm, $x=208.0$ mm, $x=270.3$ mm) are provided. The experimental data and CFD results were nondimensionalized by the jet exit conditions and ambient temperature. PIV data were acquired across the full nozzle profile, but the Raman data were only acquired near the plate surface spanning the boundary layer. Note that the y-axis range for the temperature plots are not identical and was set by the Raman data range at the measurement location. The experimental IR data are also shown on the temperature plots highlighting the steep gradient between the fluid and solid. Note that the dip in the $BR=2$ PIV velocity profile at $x=138.4$ mm and $y=42$ mm (Fig. 11) is due to a bad group of pixels on the PIV camera, resulting in erroneous velocity estimates in this region of the profile.

The RANS solutions do a reasonable job at predicting the flow profiles but underpredict the mixing in the free shear layer and boundary layer with accuracy diminishing at further aft locations. The fluid-solid and fluid-only velocity profiles are nearly identical for both blowing ratios. For BR=0, the fluid-solid results do a much better job at predicting the temperature profiles since the fluid-only simulations tend to the theoretical adiabatic wall temperature. However, for BR=2 both modeling approaches yield similar temperature predictions except very close to the plate surface. This agreement in temperature between the two approaches is the result of the interaction of the cooling flows and hot jet. Very close to the wall, it is the inner portion of the boundary layer and the CHT boundary condition on the solid surface that enables heat transfer into the plate that makes the differences observed in the surface temperature predictions over the first half of the plate. Although the flow profiles on the centerline and surface temperatures are in good agreement downstream of the third patch, the aggregate effect of the cooling holes on the entire plate temperature is poorly predicted using the CFD-only approach. From the contour plots shown in Fig. 6 – Fig. 8, it is evident that the fluid-solid method predicts the temperature distribution with better accuracy, particularly the variation across the span of the plate. The BR=1 data will be compared in the following section with the DDES results.

B. Delayed Detached Eddy Simulation CFD Analysis

RANS modeling was the focus of this work and it was determined that a fully resolved LES, modeling all of the details of the approaching boundary layers and then interaction regions between the hot boundary layer and cooling air would require computational resources in excess of what was available for this effort. However, an assessment of hybrid RANS-LES modeling was considered. The hybrid RANS-LES model utilized here is the DDES technique [16], which is an extension of the original DES technique of Spalart [17]. DDES was designed to prevent less ambiguous transition from the RANS to LES calculations for computational grids in near wall regions that were designed for RANS.

DDES simulations were considered primarily for two reasons: (1) to determine if a scale-resolved simulation would improve the prediction in both shear layers, and (2) to determine if the DDES would improve the surface temperature prediction at the end of the plate by capturing detached flow off the plate lip using LES. The interface, or “handshake region”, between RANS and LES would factor into any changes in the boundary layer prediction. The DDES simulations were run within the CFD-CHT model framework using the $k-\omega$ Menter SST model with the hybrid bounded-central differencing (BCD) discretization scheme and coupled implicit solver. The hybrid BCD scheme defaults to second order upwind in regions where RANS is applied and uses BCD in the LES regions. The converged RANS solution was used as the initial condition to the unsteady DDES simulation. A time step of 1e-6 seconds was used for this study and results were averaged for a total of 42 “plate times” where the nondimensional plate time, τ , is defined in Eq. (1).

$$\tau = \frac{U_j}{L_{plate}} t \quad (1)$$

Streamwise centerline flow profiles for BR=1 comparing fluid-only RANS, fluid-solid RANS, and fluid-solid DDES are shown in Fig. 13 (velocity) and Fig. 14 (temperature). Results for the RANS and DDES are from the fine and XFine grids, respectively. The upper shear layer prediction was improved by implementing the DDES technique. In addition, the outer portion of the boundary layer on the aft section of the plate appears to predict the flow more accurately but yields a similar prediction to the RANS results in the inner boundary layer. This shows the dependence of the DDES results on the interface region between RANS and LES, where it may be possible that with a finer grid the prediction could improve further. Although the detached flow off the plate lip was captured, very little change in surface temperature was observed.

Streamwise centerline rms profiles for both fine and XFine grid DDES cases are compared to the PIV and Raman data in Fig. 15 and Fig. 16, respectively. Nondimensional velocity and temperature rms profiles at four locations along the plate ($x=12.7$ mm, $x=138.4$ mm, $x=208.0$ mm, $x=270.3$ mm) are provided. The experimental data and CFD results were nondimensionalized by the jet exit conditions and ambient temperature. It should be noted that the rms values from these DDES computations represent only the resolved component; there is no contribution included from the modeled (or unresolved) portion that would come from the underlying RANS model. Also note that the PIV rms velocity profile at $x=12.7$ mm is close to the nozzle exit and suffers from some flare light off the surface of the nozzle.

At the first axial station in Fig. 15 ($x=12.7$ mm), there is minimal unsteady content from the solution just downstream of the nozzle exit in the upper shear layer. The DDES model is still in transition from the upstream RANS regions in the boundary layer of the upper nozzle surface and residual eddy viscosity is damping the formation of resolved turbulent structures. Further downstream, it may be observed that the resolved rms values have substantially increased to be in close agreement with the experimental data, which is due to the upper shear layer transitioning to

LES mode and resolving the unsteady turbulent motion. This is illustrated in Fig. 17, where instantaneous and mean temperature contour plots from the XFine DDES prediction are compared with the CFD-CHT RANS result. Also at the first location of $x=12.7$ mm, the DDES velocity and temperature rms values in the jet core and boundary layer are zero since the flow is attached and there is no flow unsteadiness to result in resolved turbulence. This indicates that only RANS is being applied in this region. Further downstream, the cooling flows interact with the heated jet yielding nonzero rms values indicating LES is being applied to portions of the boundary layer.

From evaluating the time-dependent temperature data throughout the domain, it was evident that the unsteady flow shedding off the nozzle lip and plate edge was captured using LES. Regarding the cooling flow interactions, the flow generally remained attached over the first patch of cooling holes; however, there was noticeable unsteadiness over the second and third patches with larger fluctuations aft of the third patch. This was observed in the time history of the cooling hole flow rates split up by patch shown in Fig. 18. The downstream patch displays the largest fluctuations with decreasing unsteadiness moving upstream. The mean flow rates out of the three patches of holes increases slightly moving downstream from the nozzle exit. This is because as the boundary layer and cooling film thicken moving downstream, the effective back pressure to each patch decreases slightly, allowing for more cooling flow mass to penetrate the flow over the plate. The same trend for the mean flow rates out of the three patches of holes was also observed in the RANS computations.

V. Conclusions

A recent film cooling experiment was conducted at NASA Glenn Research Center where a heated jet blowing over a perforated plate with three patches of cooling holes was investigated. A computational model of the film cooling experiment was built to evaluate CFD turbulence model and heat transfer techniques. A multiphysics CFD-CHT model was compared to a standard CFD-only with adiabatic wall boundary conditions approach using the $k-\omega$ Menter SST turbulence model in RANS mode. In addition, a hybrid RANS-LES analysis utilizing the DDES technique was carried out within the CFD-CHT model framework. Computational results were compared to spatially-resolved measurements of the mean and rms velocities as well as the mean and rms temperatures in the film cooling flow field. Comparisons were also made between computational results and surface temperature measurements made in the experiment using IR and thermocouple measurements.

For all three blowing ratios considered, the thermally coupled CFD-CHT results compared reasonably well to the experimental plate surface temperature data and significantly improved the prediction over the CFD-only adiabatic wall boundary condition results. The multiphysics modeling approach captured trends observed in the experimental data with the exception of the very aft end of the plate, where the CFD computed a temperature decrease while the IR data increased. The two heat transfer modeling approaches produced similar velocity and temperature flow profiles with differences observed in the temperature very close to the plate surface. Both RANS solutions underpredicted mixing in the shear layers. In general, a coupled CFD-CHT approach must be used for aerospace applications where heat transfer is important.

The DDES results improved the prediction in the upper shear layer and outer boundary layer but yielded a similar flow profile to RANS in the inner boundary layer. This was somewhat expected, because the DDES approach defaults to RANS mode in the near wall regions. Although the DDES enhanced the mixing of the hot jet with the film cooling flow on the aft end of the plate, very little change in the surface temperature was observed. The DDES model does a reasonable job predicting the velocity and temperature rms profiles and captures the trends demonstrated in the experimental data.

Acknowledgments

This work was supported by the Transformative Tools and Technologies Project of NASA's Transformative Aeronautics Concepts Program. The authors thank Jim DeBonis, Dennis Yoder, Mike Borghi, Seth Spiegel, and Mary Jo Long-Davis for helpful and insightful discussions. In addition, the authors thank Chad Winkler for his collaboration, and Amir Mansouri and Chris Nelson of Siemens Simcenter STAR-CCM+ for their insight.

References

- [1] Yoder, D. A., "Comparison of Turbulent Thermal Diffusivity and Scalar Variance Models," AIAA Paper 2016-1561, 2016.
- [2] Georgiadis, N. J., Yoder, D. A., Vyas, M. A., and Engblom, W. A., "Status of Turbulence Modeling for Hypersonic Propulsive Flowpaths," *Theoretical and Computational Fluid Dynamics*, Vol. 28, No. 3, 2014, pp. 295–318.

- [3] Yoder, D. A., DeBonis, J. R., and Georgiadis, N. J., "Modeling of Turbulent Free Shear Layer Flows," *Computers & Fluids*, Vol. 117, 2015, pp. 212-232.
- [4] Bogard, D. G., and Thole, K. A., "Gas Turbine Film Cooling," *J. of Propulsion and Power*, Vol. 21, No. 6, 2006, pp. 1-22.
- [5] Kusterer, K., Bohn, D., Sugimoto, T., and Tanaka, R., "Conjugate Heat Transfer Analysis of a Test-Configuration for a Film-Cooled Blade," *Proceedings of the International Gas Turbine Congress 2003 Tokyo*, IGTC2003Tokyo TS-083, 2003.
- [6] Borghi, M. R., Thurman, D., Poinsette, P., Wernet, M., and Engblom, W.A., "Preliminary Numerical and Experimental Evaluation of Turbulent Mixing from Thin-Film Cooling," *AIAA Paper 2017-1004*, 2017.
- [7] Bridges, J. and Wernet, M. P., "The NASA Subsonic Jet Particle Image Velocimetry (PIV) Dataset," *NASA Technical Memorandum*, 2011, NASA/TM-2011-216807.
- [8] Locke, R. J., Wernet, M. P., Anderson, R. C., "Rotational Raman-Based Temperature Measurements in a High-Velocity Turbulent Jet," *Meas. Sci. Technol.*, Vol. 29, No. 1, Dec. 2017, pp. 1-16.
- [9] Wernet, M. P., Georgiadis, N.J., and Locke, R.J., "PIV and Rotational Raman-Based Temperature Measurements for CFD Validation in a Single Injector Cooling Flow," *AIAA Aviation 2018 paper* and also published as NASA/TM—2018-3857, June 2018.
- [10] Wernet, M.P., Georgiadis, N.J., Locke, R.J., Thurman, D.R., and Poinsette, P.E., "PIV and Rotational Raman-Based Temperature Measurements for CFD Validation of a Perforated Cooling Flow – Part 1", *AIAA Paper 2020-1230*, Jan. 2020.
- [11] Georgiadis, N. J., Yoder, D. A., and Engblom, W. A., "Evaluation of Modified Two-Equation Turbulence Models for Jet Flow Predictions," *AIAA Journal*, Vol. 44, No. 12, 2006, pp. 3107-3114.
- [12] Klopp, W. D., *Aerospace Structural Metals Handbook*, Ferrous Alloys, Type 316 & 317, Code 1307, June 1988.
- [13] Simcenter STAR-CCM+, Software Package, Version 15.02.007-R8, Siemens Digital Industries Software, Plano, TX, 2020.
- [14] Menter, F. R., "Improved Two Equation $k-\omega$ Turbulence Models for Aerodynamic Flows," *NASA/TM 103975*, October 1992.
- [15] Menter, F. R., "Two Equation Eddy-Viscosity Turbulence Models for Engineering Applications," *AIAA Journal*, Vol. 32, No. 8, 1994, pp. 1598-1605.
- [16] Trapier, S., Deck, S., and Duveau, P., "Delayed Detached-Eddy Simulation and Analysis of Supersonic Inlet Buzz," *AIAA Journal*, Vol. 46, No. 1, 2008, pp. 118-131.
- [17] Spalart, P.R., "Detached Eddy Simulation," *Annual Review of Fluid Mechanics*, Vol. 41, 2009, pp. 181-222.

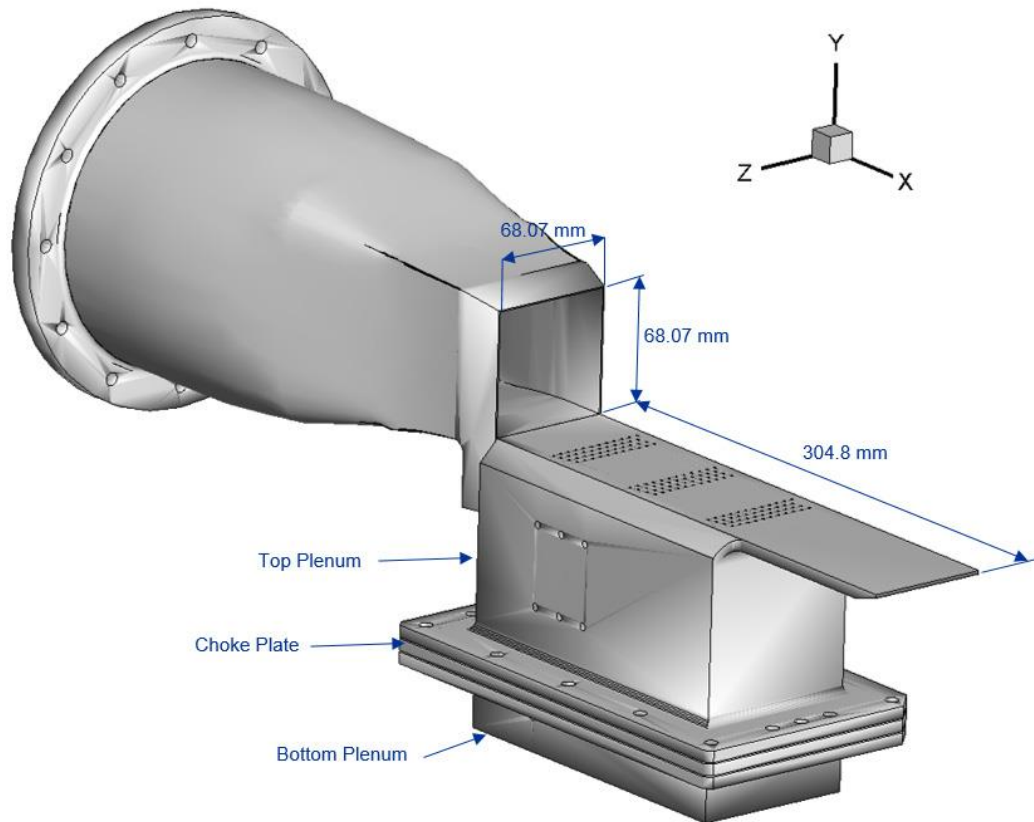


Fig. 1 Experimental configuration - square nozzle with 6.35 mm thick plate and three patches of forty-five 1 mm (drilled to 0.040 inch) diameter injector holes angled at 30° to plate surface. Cooling air is provided by the two chamber plenum.

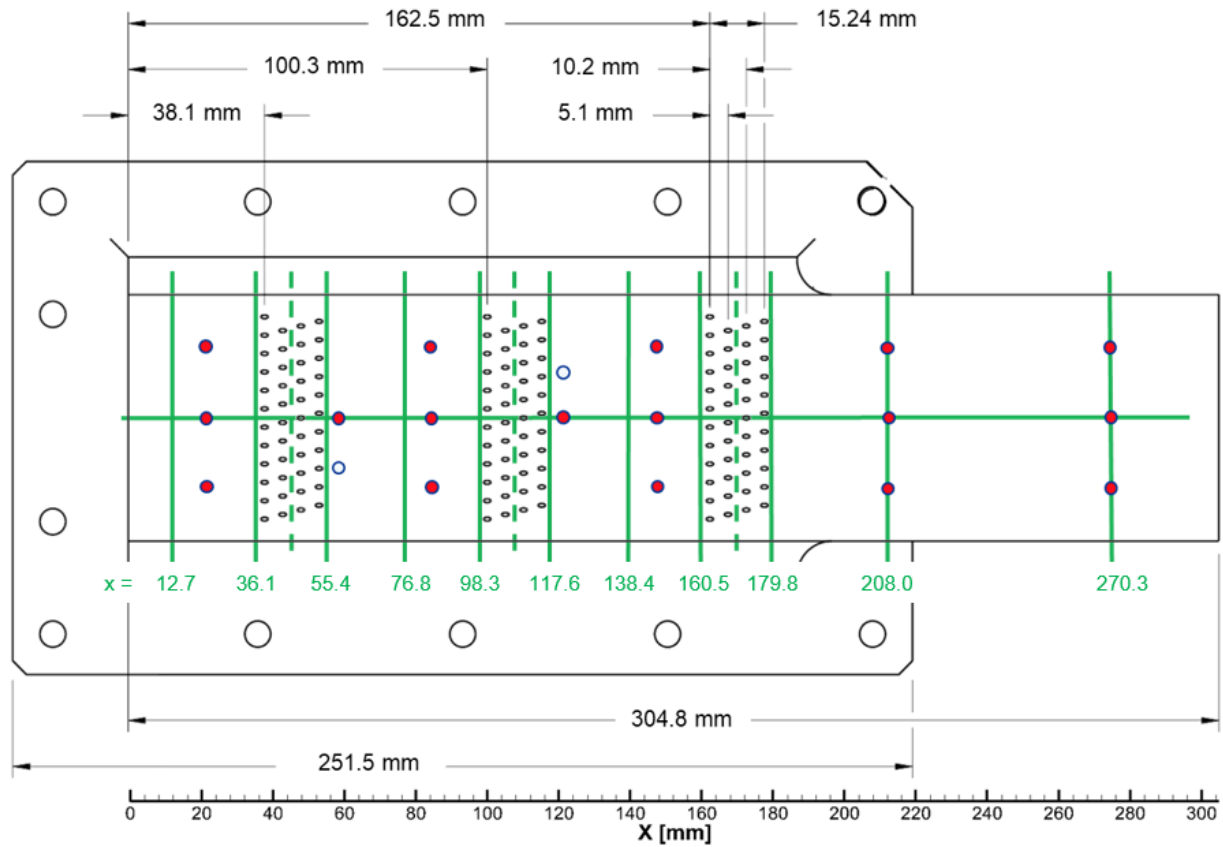


Fig. 2 Top view of the plate showing the cooling hole geometry and dimensions. The scale on the bottom provides a reference for the axial locations of the Raman, Thermocouple, and PIV measurements.

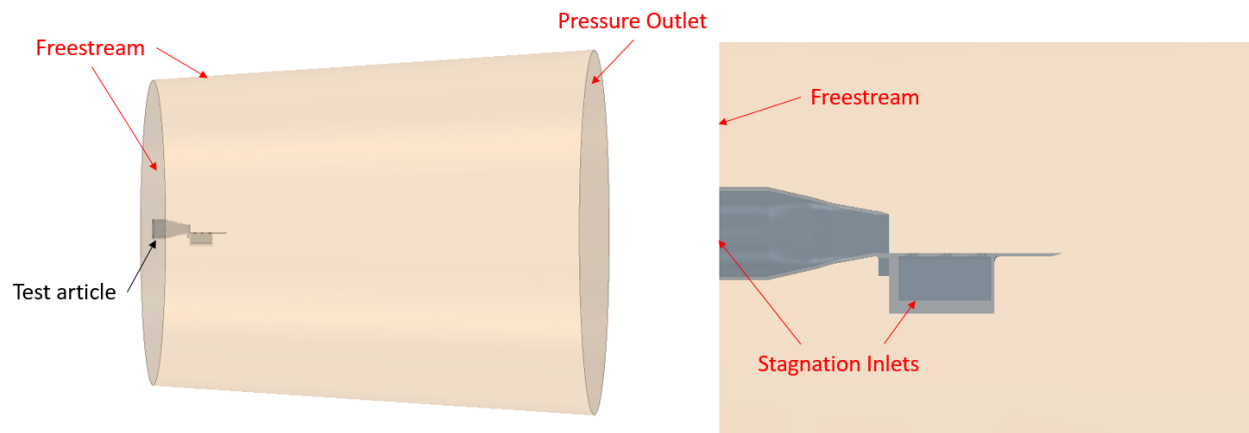


Fig. 3 Computational domain and boundary conditions showing full domain (left) and zoomed-in slice (right).

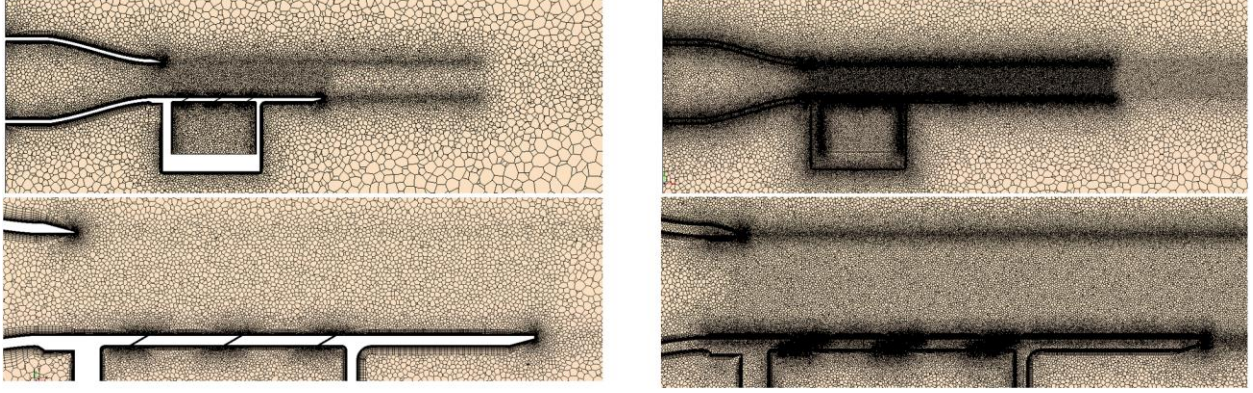


Fig. 4 Comparison of fluid-only coarse grid (left) and fluid-solid fine grid (right) on nozzle centerline plane.

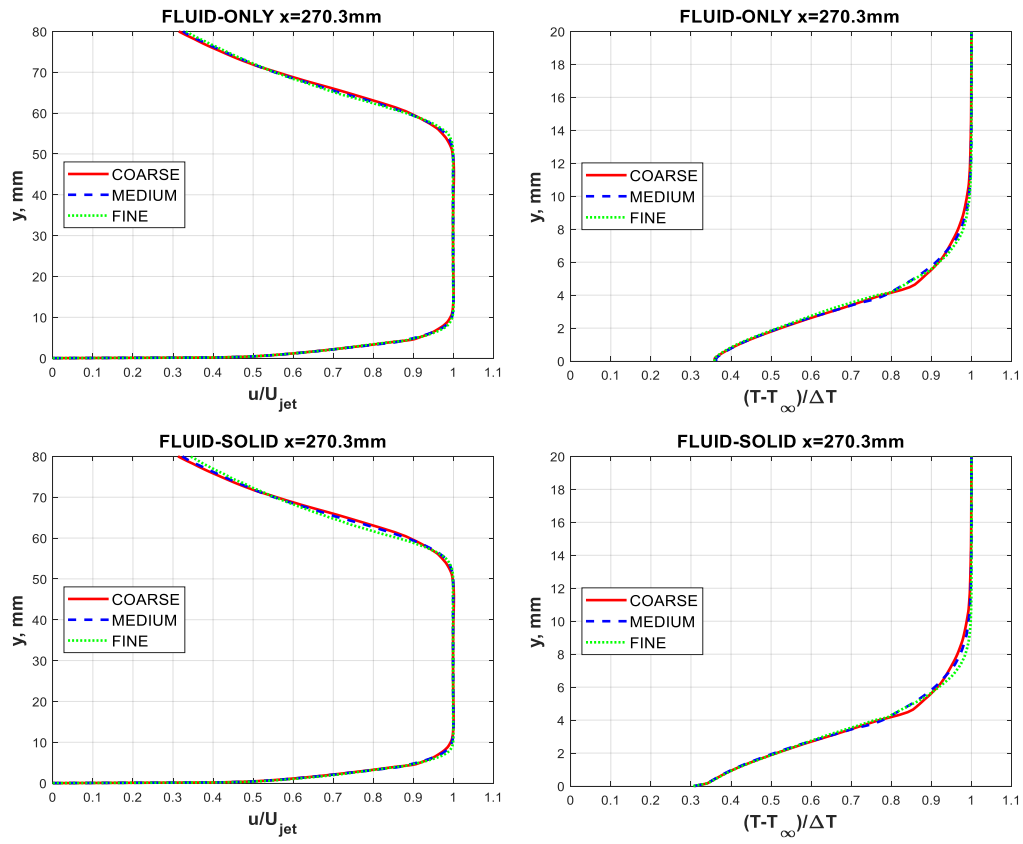


Fig. 5 Grid independence study for fluid-only (top) and fluid-solid (bottom) grid families showing streamwise centerline axial velocity (left) and temperature (right) flow profiles at $x=270.3$ mm for $BR=1$.

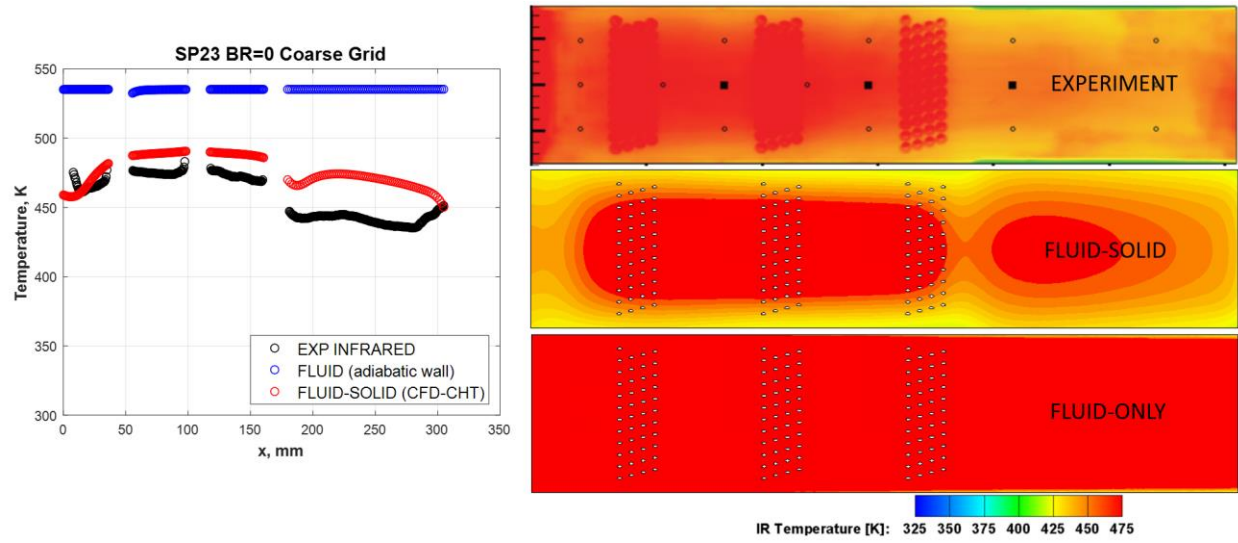


Fig. 6 Centerline surface temperature (left) and plate surface temperature contours (right) for BR=0.

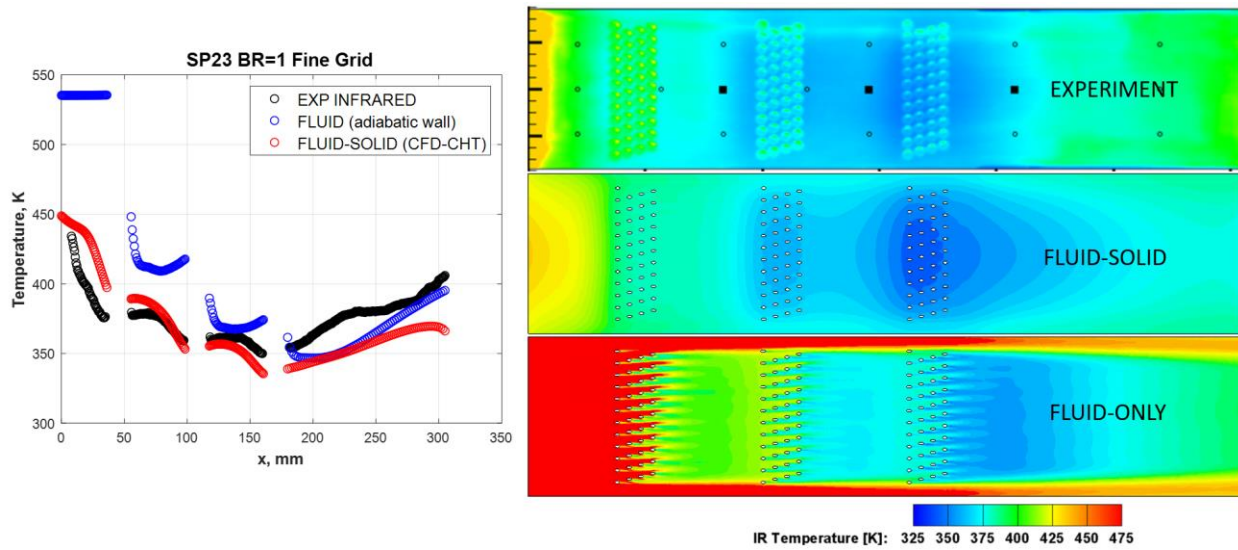


Fig. 7 Centerline surface temperature (left) and plate surface temperature contours (right) for BR=1.

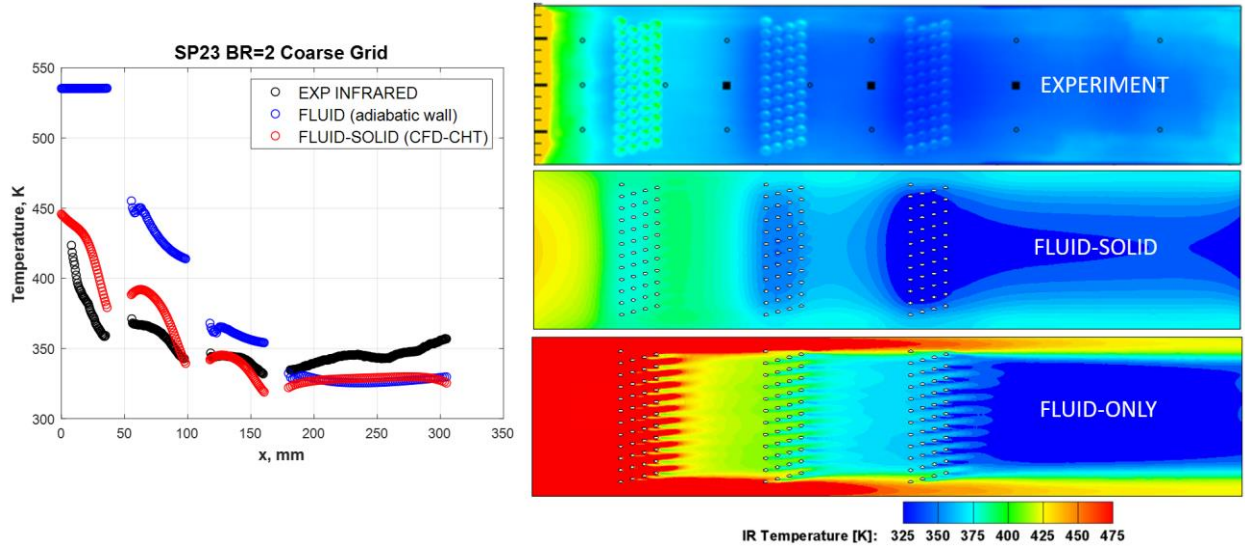


Fig. 8 Centerline surface temperature (left) and plate surface temperature contours (right) for BR=2.

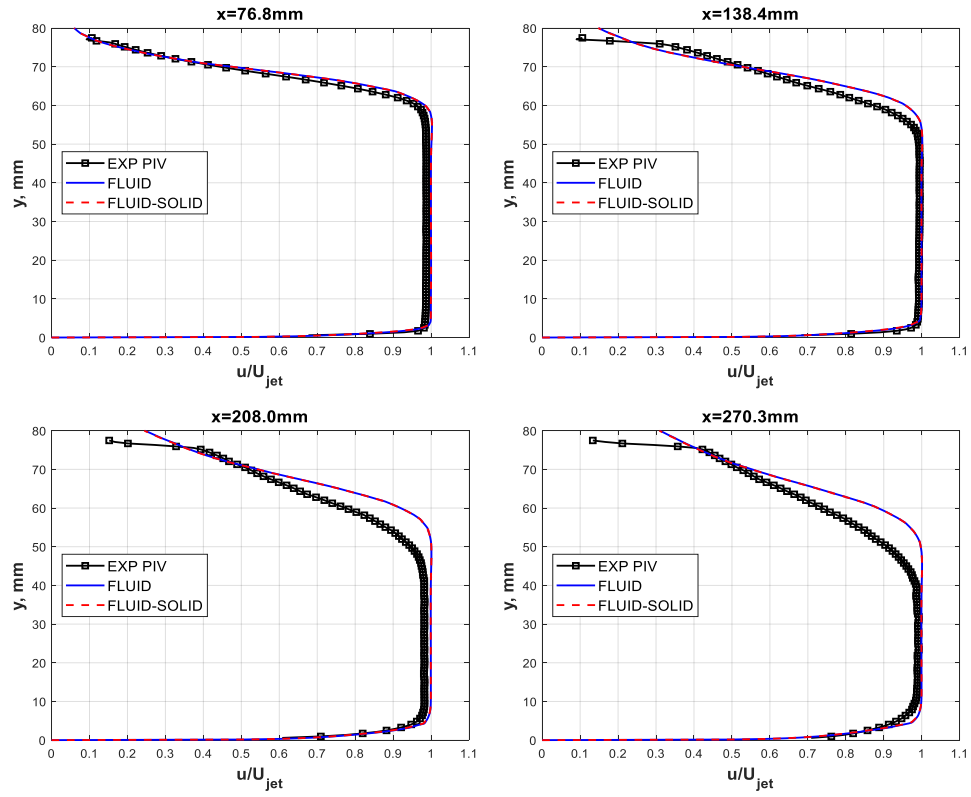


Fig. 9 Streamwise centerline mean velocity profiles for BR=0.

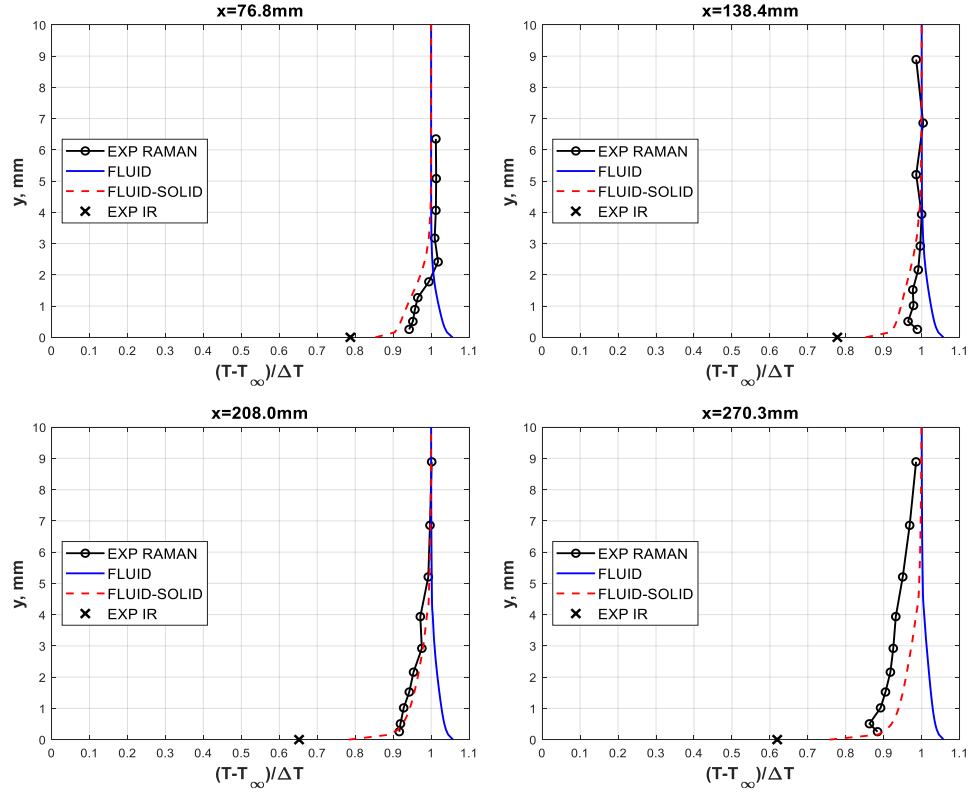


Fig. 10 Streamwise centerline mean temperature profiles for BR=0.

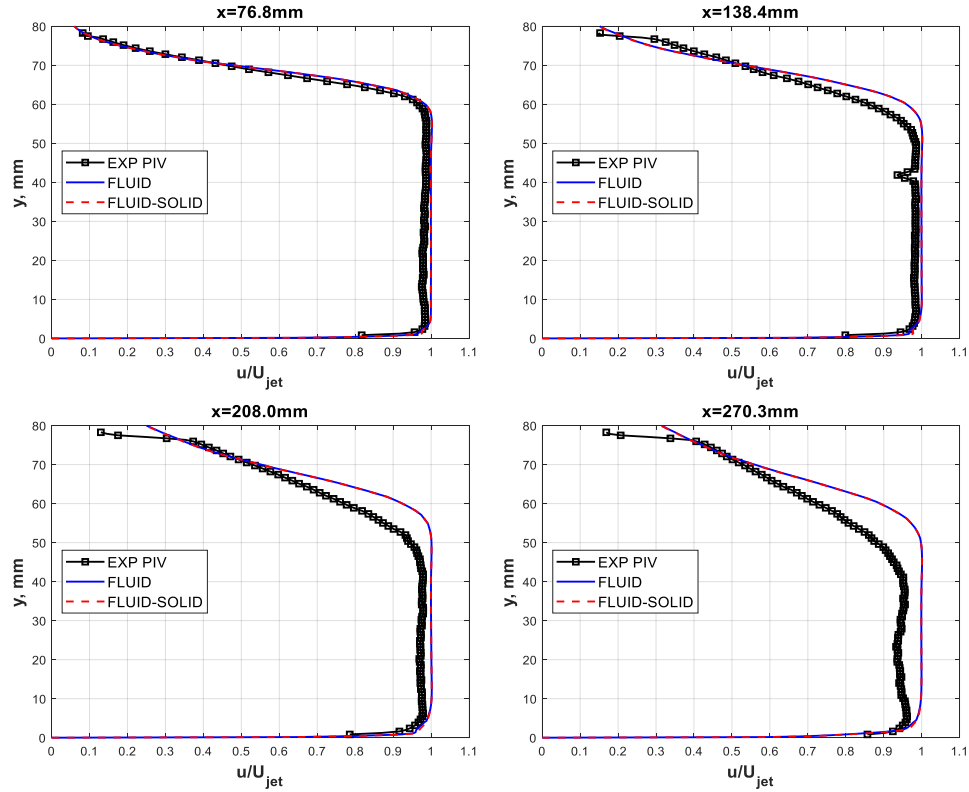


Fig. 11 Streamwise centerline mean velocity profiles for BR=2.

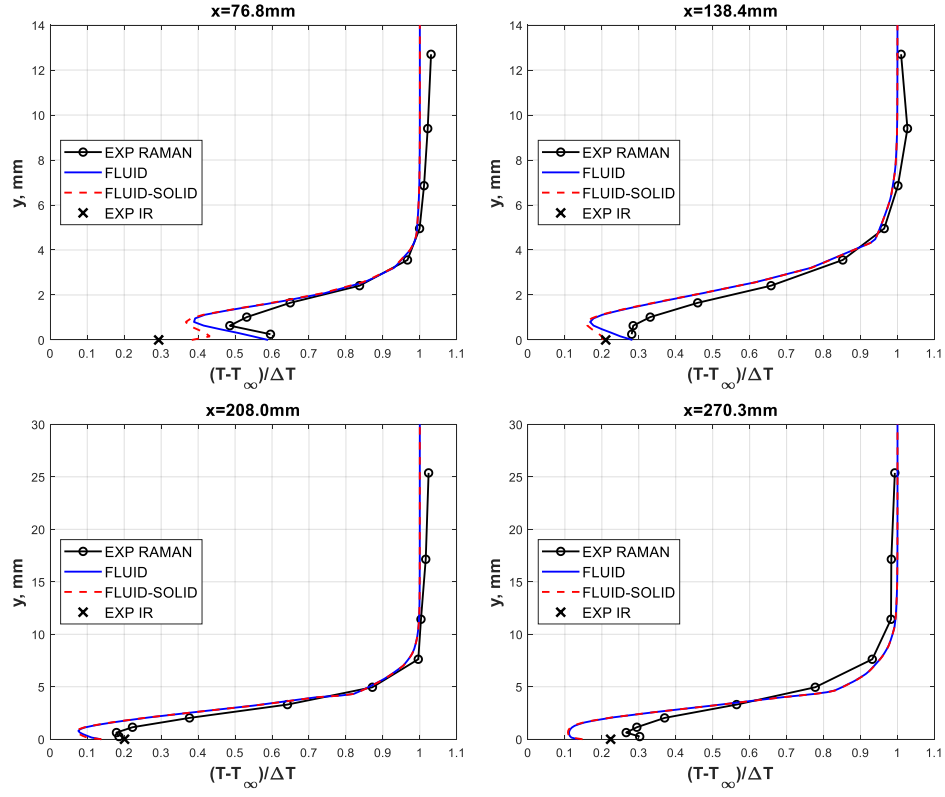


Fig. 12 Streamwise centerline mean temperature profiles for BR=2.

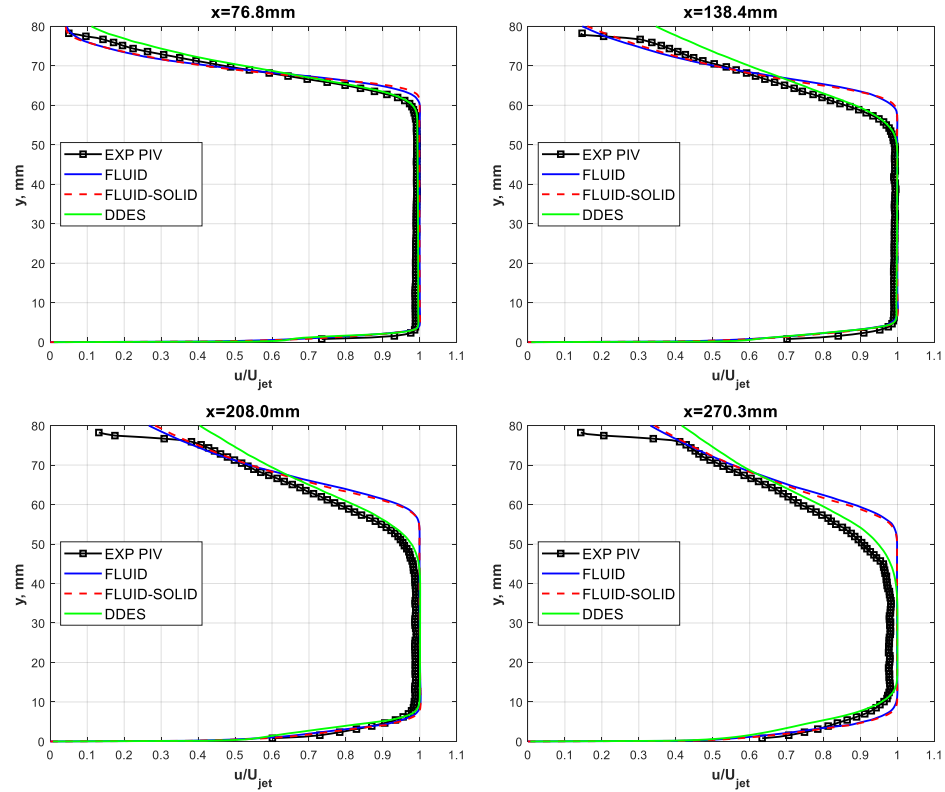


Fig. 13 Streamwise centerline mean velocity profiles for BR=1.

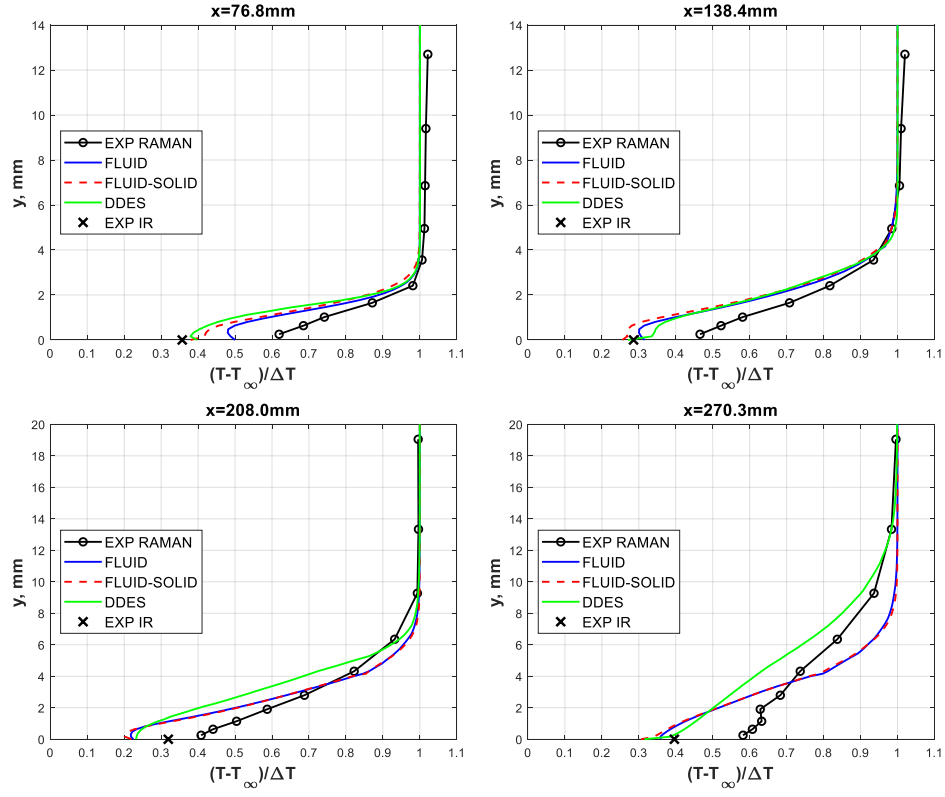


Fig. 14 Streamwise centerline mean temperature profiles for BR=1.

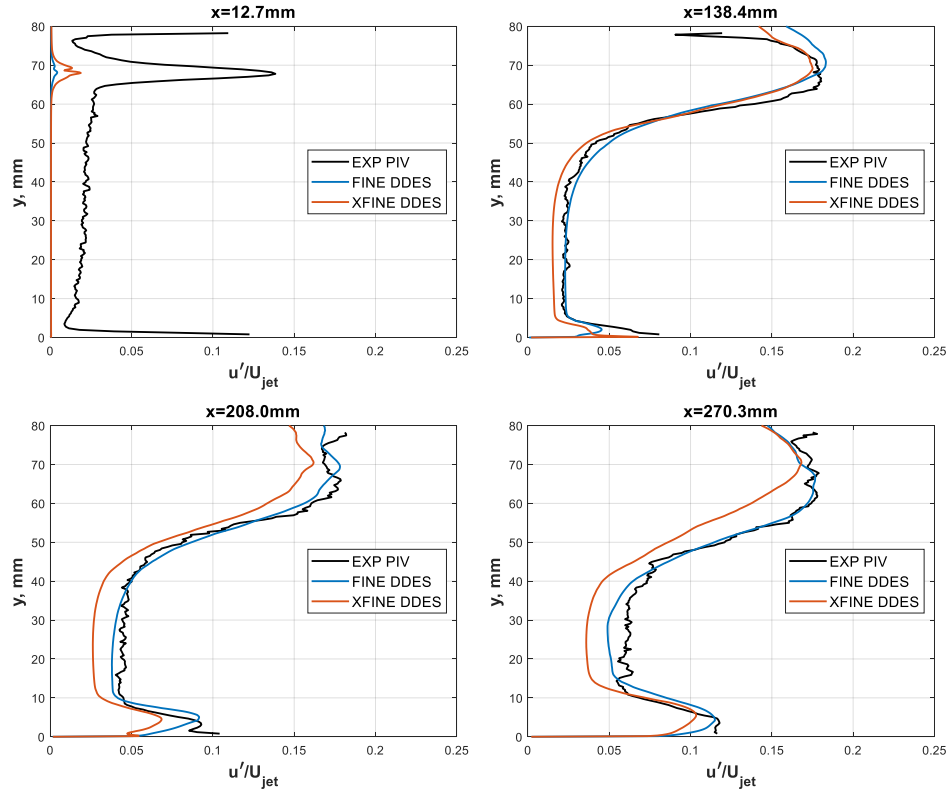


Fig. 15 Streamwise centerline rms velocity profiles for BR=1.

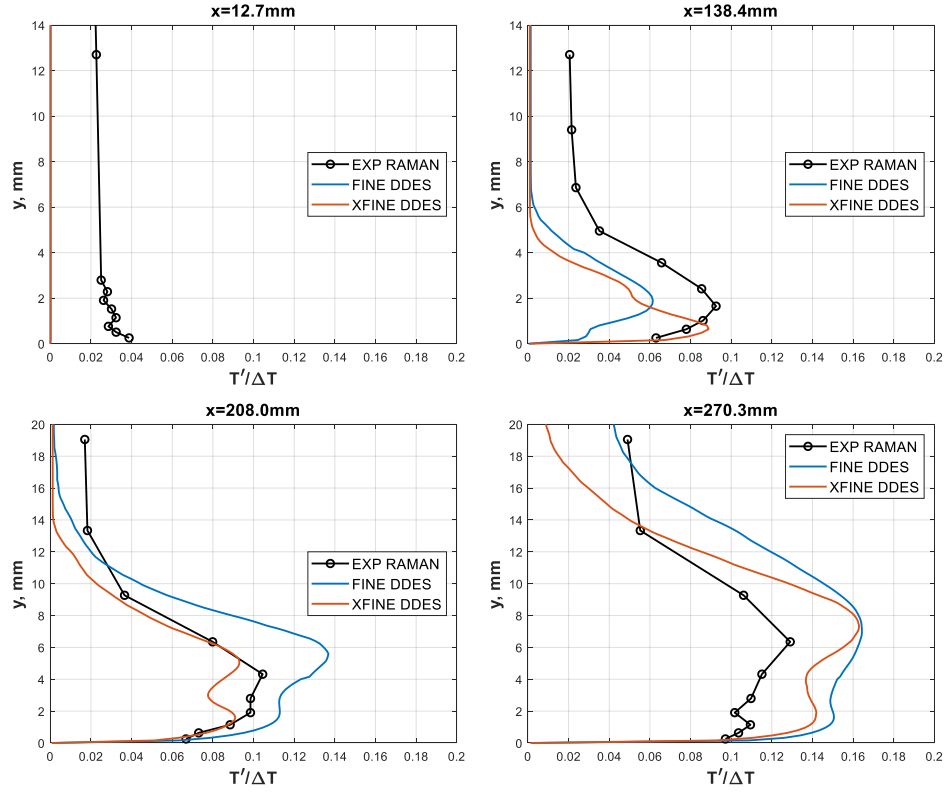


Fig. 16 Streamwise centerline rms temperature profiles for $BR=1$.

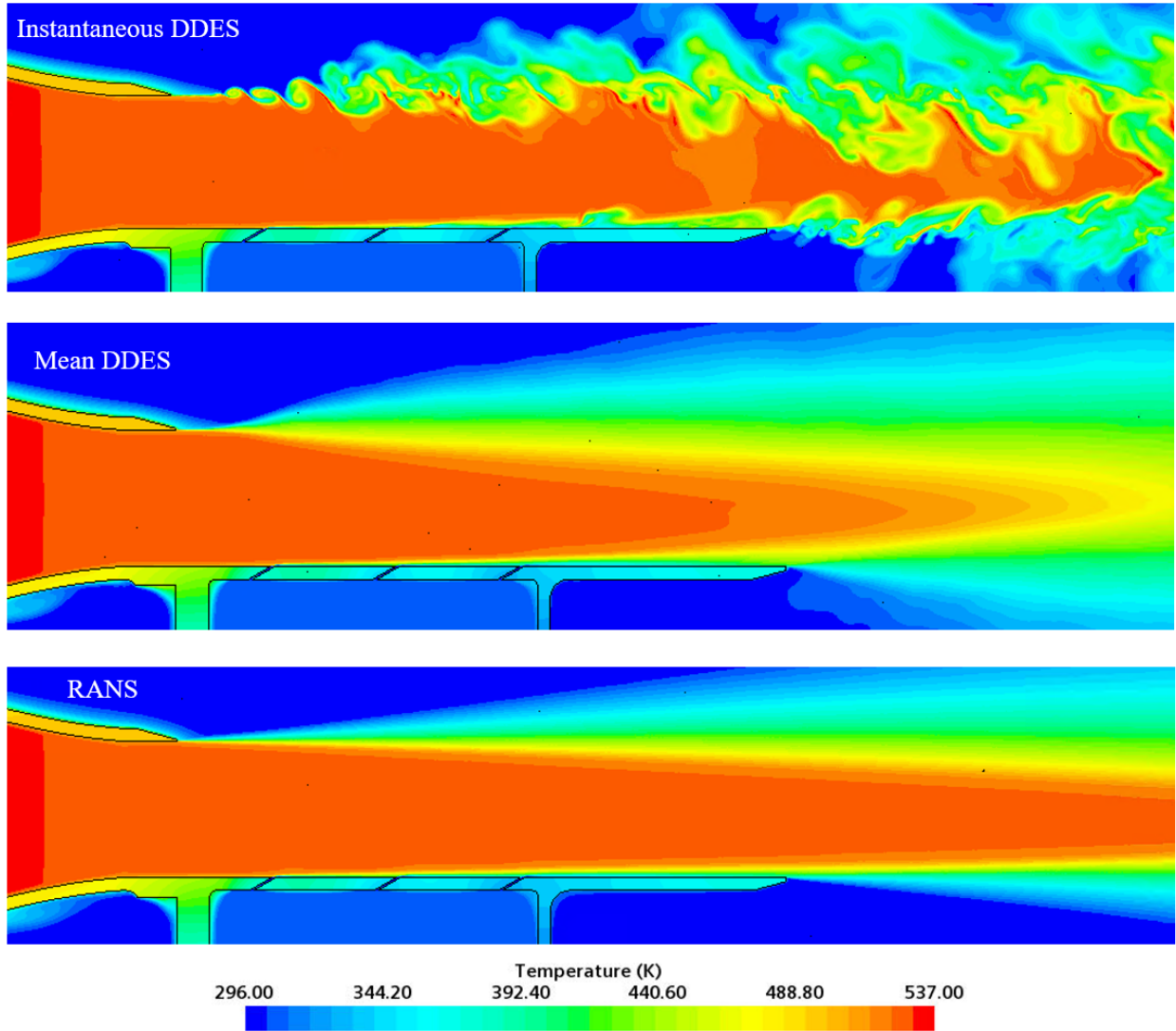


Fig. 17 Temperature contours of instantaneous DDES (top), mean DDES (middle), and RANS (bottom) results on nozzle centerline plane for $BR=1$.

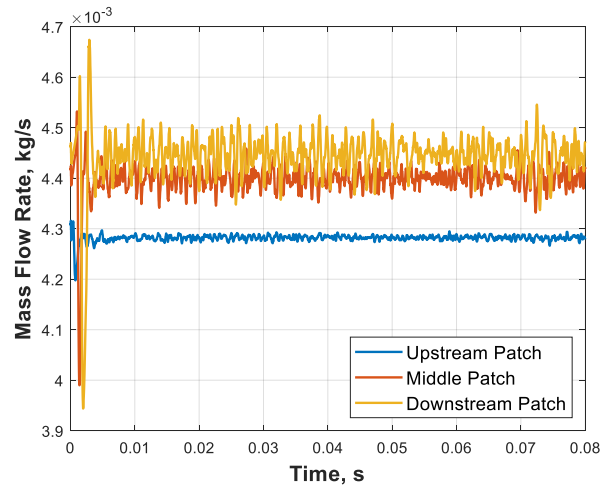


Fig. 18 Time history of cooling hole flow rate by patch from XFine DDES computation for BR=1.