JOURNAL OF THERMOPHYSICS AND HEAT TRANSFER
Vol. 13, No. 2, April–June 1999

Aircraft Skin-Cooling Airflow Distribution

Julie Schneider,* Iran Spradley,* and Ab Hashemi*
Lockheed Martin Advanced Technology Center, Palo Alto, California 94304-1191
Jay Nigen†
Innovative Research, Inc., Kanata, Ontario K2K 2M8, Canada
and
Elizabeth Dyson‡
Lockheed Martin Canada, Kanata, Ontario K2K 2M8, Canada

A skin-cooling system has been developed to reject heat from high-power electronic equipment onboard a CL-600 Challenger aircraft. This concept calls for baffled ducts at the aircraft skin to cool the air in the aft cabin, where the equipment is located. The hot air from the rack with the equipment that dissipates the most heat is exhausted into a ceiling plenum, from which it is distributed by fans into the baffled ducts. These ducts, which are bounded by the cold skin on one side, are used as a heat exchanger to reject the heat. The cooled air is then recirculated into the aft cabin. This paper describes a series of tests that were performed to evaluate the feasibility of providing a common inhale/exhale opening in the ceiling plenum. This opening is to prevent the ceiling fans from starving when some of the equipment is off (inhale), and to avoid excessive pressurization of the system when some or all of the ceiling fans are off (exhale). The tests were performed using a representative plenum configuration. This paper also describes a simulation model of the plenum flow distribution system and comparison of the predicted result with the experimental measurements. Results show that a common inhale/exhale opening in the ceiling plenum is feasible for the aircraft supplementary cooling system. A substantially larger opening area in the plenum is required for the exhale mode of operation than is needed for the inhale scenario. Flow deflectors near the openings are required to prevent bidirectional flow through the openings during a balanced flow mode of operation. Bidirectional flow causes some of the hot air to mix with the aft cabin air prior to being cooled by the cold skin. This reduces the effectiveness of the heat-rejection system. The results also include the description of tested measurement techniques and selection of devices to optimize the size of the inhale/exhale opening.

Nomenclature

A = link cross-sectional area
$F(\dot{Q})$ = additional head from fans, pumps, blowers, etc.
h = specific enthalpy
$L(\dot{Q})$ = flow losses associated with either friction or other effects
$\dot{M}$ = mass generation
P = total pressure
$\dot{Q}$ = volumetric flow rate
$\dot{S}$ = energy generation
t = time
V = volume
$\dot{W}$ = energy generation associated with compressibility effects
ρ = density

Introduction

RETROFITTING of existing military and commercial aircraft with high-power electronic equipment provides an additional heat source in the cabin. The existing aircraft environmental control system cannot handle this additional heat load; thus, the aircraft requires supplemental cooling to fulfill its mission.

To overcome this problem in a CL-600 Challenger aircraft, a skin-cooling design has been developed. The design is fully described in Refs. 1–4. The concept calls for baffled ducts at the aircraft skin to cool the air in the aft cabin, where the equipment is located. The hot air from the rack with the equipment that dissipates the most heat is exhausted into a ceiling plenum, from which it is distributed by fans into the baffled ducts. These ducts, which are bounded by the cold skin on one side, are used as a heat exchanger to reject the heat. The cooled air is then recirculated into the aft cabin. A configuration of the ceiling plenum is shown in Fig. 1.

The overall objective of the effort described in this paper is to experimentally evaluate the feasibility of providing a common inhale/exhale opening in the ceiling plenum. This opening is to prevent the ceiling fans from starving when some of the rack equipment is off (inhale), and to avoid excessive pressurization of the rack plenum when some or all of the ceiling fans are off (exhale). The technical approach was to modify the test section of an existing wind tunnel to represent half of a long section of the ceiling plenum with inhale/exhale openings, and to experimentally characterize the flow through these openings. Then, a simulation model of the system was developed using MacroFlow[5] computer software, and the experimental results were compared with the theoretical predictions.

Four fans were used to simulate the operation of the ceiling fans in the supplementary cooling system. The size of the inhale/exhale openings was varied to investigate the feasibility of using the same opening for both inhale and exhale modes of operation without substantial flow reversal through or pressure drop across the openings in either mode. The tests established the feasibility of using a common opening for both inhale and exhale modes of operation.

Presented as Paper 98-0839 at the AIAA 36th Aerospace Sciences Meeting, Reno, NV, Jan 12–15, 1998; received March 13, 1998; revision received Nov. 10, 1998; accepted for publication Nov. 10, 1998.

*Missiles and Space, Thermal Sciences.
†Currently with Fluent, Inc., Santa Clara, CA 95054.
‡Currently with Systemhouse, Hull, Quebec J8X 4B7, Canada.

Experimental Setup

A schematic of the wind tunnel is shown in Fig. 2. The wind tunnel is a blow-through type open-circuit facility. The approximate overall dimensions are 20 ft long, 6 ft high, and 6 ft wide (excluding instrument traverse, described later). The tunnel uses a NYB Acoustafoil Blower (no. 279) to supply air. The blower is nominally rated at 14,000 cfm, but actual performance because of pressure drops in the facility is ~11,000 cfm. Guards are provided for the blower inlet, shaft, and drive belts. The blower is powered by a 5-hp Eaton Dynamatic motor with an eddy current clutch controller to allow repeatable and accurate speed setting. The facility power is from a 208/220-V three-phase source.

The contraction and exit diffuser are constructed of 14 gauge 304 stainless steel. The profile of the contraction section is based on a fifth-order polynomial with 5.76:1 area ratio. The inlet and outlet of each section are reinforced by $\frac{1}{4}$-in. stainless-steel flanges. Perforated metal plates are placed at the inlet and outlet of the diffuser to help mitigate separation and straighten the blower flow. A $\frac{1}{4}$-in. cell-size aluminum honeycomb and three fine-mesh stainless-steel screens condition and homogenize the airflow prior to entering the contraction section. The honeycomb and screens are mounted in hardwood frames.

The test section of the wind tunnel was modified to form a 6-ft-long rounded-top 6061-T6 aluminum duct. This represents half of the long section of the ceiling plenum onboard the aircraft, including the curvature of the airplane's cabin, and the fans used to withdraw air from the plenum. Only half of the long section of the aircraft plenum was modeled because of the symmetry of this section, and it was consistent with the objective of the tests to characterize rather than to quantify the flow. Because the tests were performed under isothermal conditions, buoyancy did not play a role in flow distribution. Therefore, for convenience, the test section was inverted so that the inhale/exhale openings faced upward in the laboratory setup. A schematic of the modified test section with the inhale/exhale openings is shown in Fig. 3. The overall dimensions of the duct cross section are 14.5 in. by 13.5 in. The test section is 6 ft. long and 15 in. square in cross section. The test-section walls are clear anodized, whereas the remaining parts are black anodized. Scratch-resistant Plexiglas® windows in easily removable frames allow access to the test section. Inhale/exhale openings were provided in the top plate of the test section, opposite the rounded surface, and fans were located on the side perpendicular to this surface. Five access ports and sealing plugs were provided in the top plate of the test section.

Four off-the-shelf exhaust fans (FANTECH FR150) were used to simulate the cooling system fans. These fans had the same flow throughput as the aircraft fans. Modifications to the test setup were made to simulate the as-installed configuration of the actual fans as closely as possible.

During the shakedown tests, flow reversal was observed through the holes during exhale or balanced flow modes of operation. This flow reversal causes some of the hot air to mix with the aft cabin air prior to being cooled by the cold skin, which reduces the effectiveness of the heat-rejection system.

This problem was resolved by installing airflow deflectors upstream of each hole. A schematic of the airflow deflectors is shown in Fig. 4. The shape of the deflectors was semicircle with a radius of about 1 in. The deflectors cause the flow to separate from the wall, pass over the holes, and then reattach to the wall.

Flow measurements were performed using a 5.75-in. i.d. by 64-in.-long tube and Alnor Velometer (series 6000P, 0-1250 and 0–2500 ft/min). Pressure measurements were made using a Dwyer Manometer (model no. 400, 0–10 in. of H_2O). Because of the turbulent and rotational nature of the airflow out of the exhaust fans, the 5.75-in. i.d. tube was modified. Eight 1 in. × 18 in. long baffles were installed in the upstream end of the tube to help straighten the flow and allow uniform flow measurements. A schematic of the airflow measurement system is shown in Fig. 5. Telltales were used for flow visualization.

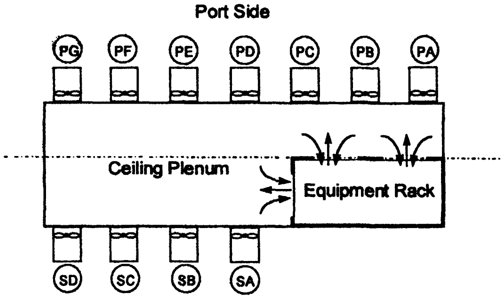

Fig. 1 Schematic of the ceiling plenum.

Experiments

Initially, the characteristics of a single Fantech FR150 fan were determined. The volumetric flow output of the fan as a function of pressure drop across the fan was measured as shown in Fig. 6. The pressure drop across the fan was controlled using a flapper valve within the 5.75-in. i.d. flow tube. Nonuniform velocities were measured around the circumference of the exit area of the 5.75-in. i.d. flow tube. Airflow was measured several times at the same location to reduce the uncertainty in the measurement.

Inhale Pressure Drop

The first series of tests was performed on the inhale configuration that allows the plenum to breathe in through the bypass holes when the inlet supply of air is restricted or turned

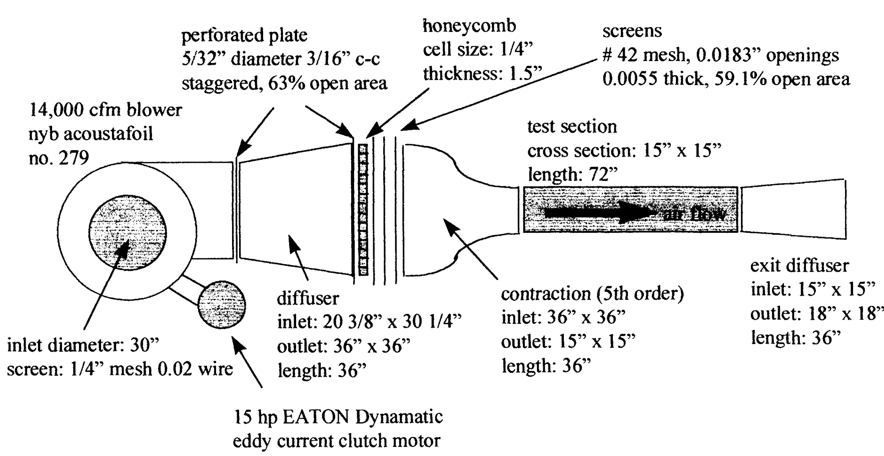

Fig. 2 Modified wind tunnel.

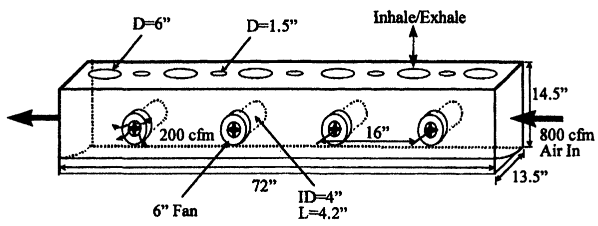

Fig. 3 Inhale/exhale test section.

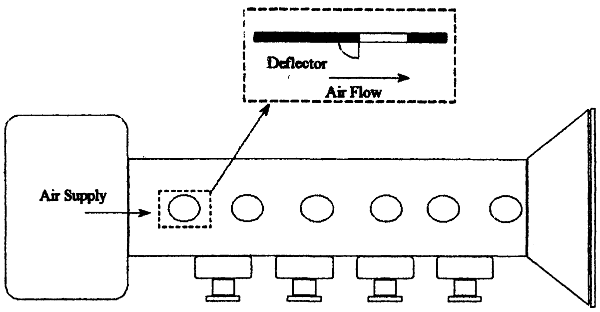

Fig. 4 Airflow deflector.

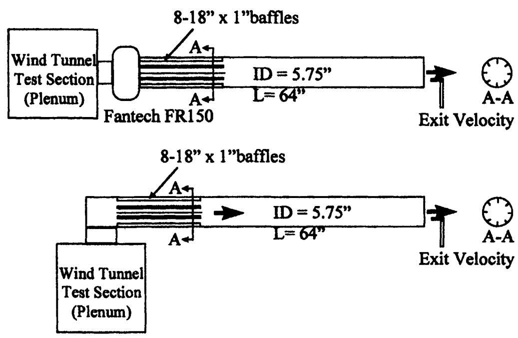

Fig. 5 Airflow measurement system.

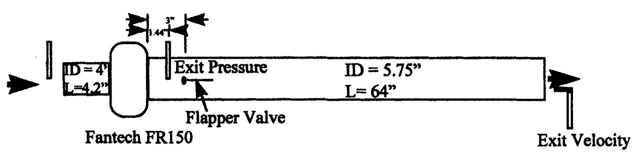

Fig. 6 Initial exhaust fan flow characterization test setup.

off. This configuration is shown in Fig. 7. Three different bypass areas were investigated for this configuration. They included 0.196 ft^2 (one, 6-in. hole open), 0.59 ft^2 (three, 6-in. holes open), and 1.18 ft^2 (six, 6-in. holes open). The airflow rate through the fans was varied between 0 and 800 cfm, i.e., one fan on, two fans on, etc. Flow rates were measured from each exhaust fan using the 5.75-in. i.d. flow tube and Alnor Velometer. Flow rates ranged from 0 to ~850 cfm. Pressure change within the plenum was measured using a Dwyer Manometer with the two ends of the manometer reversed from the exhale configuration to account for the negative pressure change.

Exhale Pressure Drop

The second series of tests was performed on the exhale configuration that allows the plenum to breathe out through the bypass holes when the exhaust fans are turned off or have insufficient throughput to handle the inlet flow of air. This configuration is shown in Fig. 8. Four different openings were investigated for this configuration. The opening areas were 0.59 ft^2 (three, 6-in. holes open), 0.787 ft^2 (four, 6-in. holes open), 1.18 ft^2 (six, 6-in. holes open), and 1.6 ft^2 (six, 6-in. holes open, and exit rectangular, 4.25 in. x 14.25 in. area added at end of diffuser section of the wind tunnel). Tests with smaller exhale openings, i.e., one or two holes, were not performed because they caused undesirably high-pressure buildup (>0.5-in. w.g.) in the plenum. The supply airflow rate was varied between 0 and 800 cfm.

Flow rate measurements at the extremely low end (0–200 ft/min) were difficult to determine because of the precision of the Alnor Velometer within this range.

Inhale Fan Sequencing

The third series of tests was performed to simulate the effect of inhale operational scenarios on plenum pressure. Inhale scenarios were performed with 0-cfm inlet flow and the exhaust fans turned on in sequence. The exhaust fans were turned on in various combinations with the entire possible number of combinations explored. A typical test configuration is shown in Fig. 9.

Exhale Fan Sequencing

The fourth series of tests was performed to simulate the effect of reduced supply inlet flow on plenum pressure. 200-, 400-, 600-, and 800-cfm inlet flows were provided with the blower and removed from the plenum test chamber by the exhaust fans. The exhaust fans were turned off in various combinations with the entire possible number of combinations. A typical test configuration is shown in Fig. 10.

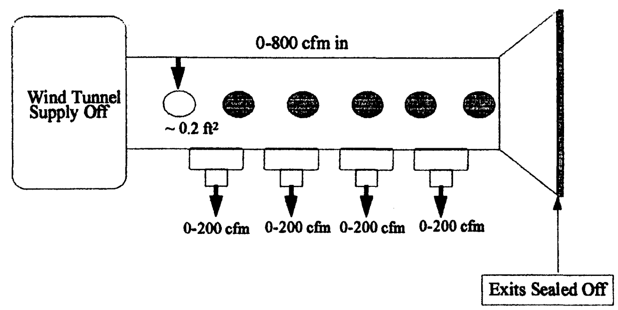

Fig. 7 Inhale test configuration.

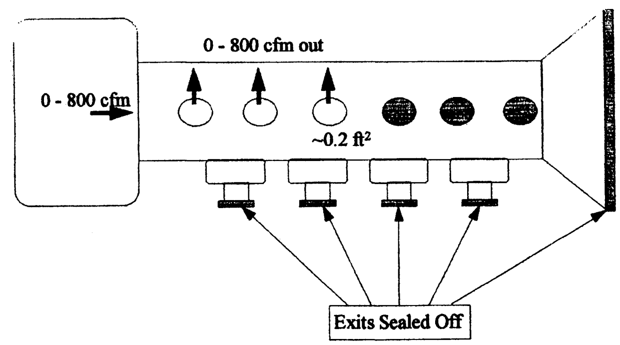

Fig. 8 Exhale test configuration.

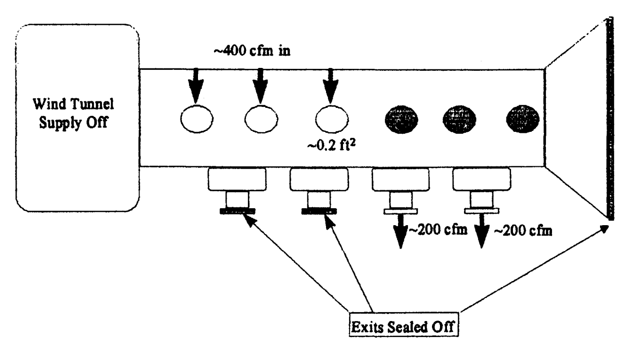

Fig. 9 Typical inhale operational scenario.

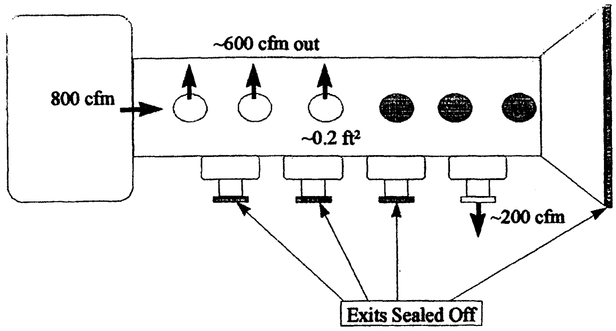

Fig. 10 Typical exhale operational scenario.

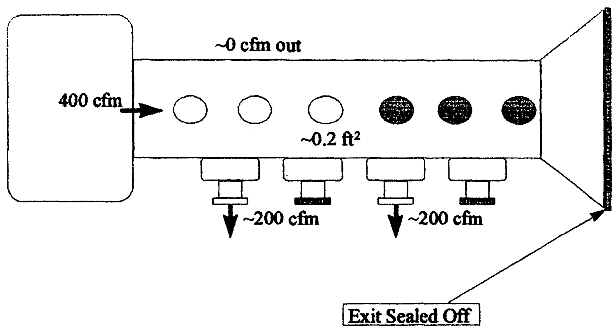

Fig. 11 Typical balanced flow configuration.

Balanced Flow Fan Sequencing

The fifth and final series of tests was performed to characterize system performance during balanced flow mode of operation, i.e., amount of air removed by fans was equal to the air supplied by the blower. Again, 200-, 400-, 600-, and 800-cfm inlet flows were provided with the blower and removed from the plenum test chamber by the exhaust fans. The exhaust fans were turned off in various combinations, while maintaining balanced flow, with the entire possible number of combinations. A typical test configuration is shown in Fig. 11.

To prevent reverse flow through the holes, the flow deflectors were a necessity during this mode of operation. To fully investigate the reverse-flow phenomena, a test with all six holes opened was also performed. The flow rate for this test was 800 cfm and all of the fans were turned on. With the air deflectors designed for three-hole opening, null flow could not be achieved through the six holes. In this configuration, additional air was still being sucked into the plenum through the holes closest to the inlet supply, and exhausted out of the plenum through the holes farthest from the inlet supply.

Modeling and Simulation

A simulation model of the experimental setup was developed using the MacroFlow[5] computer code. For completeness, a general description of MacroFlow is presented next, and then the specific model for the experimental setup is described.

In MacroFlow, network representations of flow systems are constructed from a library of components, tees, crosses, elbows, orifices, valves, etc., and connections, pipes, logical connections, fixed flow rate, etc. After the network is constructed, the user specifies any required component or link specific information and then runs the simulation. Within the computational engine, each component is mathematically modeled by a combination of two fundamental building blocks, the link and the node. Then, the conservation of mass, momentum, and energy for this link-node-based mathematical abstraction of the physical system are expressed and solved. State variables, such as pressure and temperature, as well as physical properties are stored and solved for each node. At a link, flow rate and convective fluxes are stored and solved. This link-node-based methodology is equivalent to the use of a staggered grid arrangement, which is employed in many computational fluid dynamics formulations.

The integrated forms of mass, momentum, and energy conservation applicable to a link-node network are described next. In these equations, quantities with two subscripts, e.g., Q_{ij}, are associated with links, whereas those with a single subscript, e.g., p_i, are associated with nodes.

Mass conservation is applied at each node. Thus, the mass balance equation for node i can be represented as follows:

$$\frac{\partial(\rho_i V)}{\partial t} = \sum_{k=1}^{n_{ij}} (\rho_{ij}\dot{Q}_{ij})_k + \dot{M}_i$$

Momentum conservation is formulated at each link. Although multiple links can connect to a single node, only two nodes can be connected to a single link. Conservation of momentum expressed for a link is as follows:

$$\frac{V_{ij}}{A_{ij}}\frac{\partial(\rho_{ij}\dot{Q}_{ij})}{\partial t} = \Delta P_{ij} - L(\dot{Q}_{ij}) + F(\dot{Q}_{ij})$$

The second term on the right-hand side (RHS) of the preceding equation accounts for losses associated with any flow resistances. Such flow resistances include viscous losses (shear) and minor losses, produced by flow separation. Minor losses reflect that momentum from the streaming flow is consumed to drive the recirculating zones or eddies present in separated flow regions. Traditionally, minor losses are expressed in terms of k factors, valve coefficients, or simply total pressure drop. The final term on the RHS represents the additional driving force provided by any fans, blowers, or pumps within the link.

As with mass conservation, energy conservation is expressed at each node. Rather than formulating the conservation of energy with temperature as the primary variable, enthalpy is selected to remove any difficulties associated with temperature-dependent specific heat:

$$\frac{\partial(\rho_i V_i h_i)}{\partial t} = \sum_{k=1}^{n_{ij}} (\rho_i \dot{Q}_{ij} h_{ij})_k + \dot{S}_i + \dot{W}_i$$

Because the experiments were performed under isothermal conditions, the energy equation was not solved in this application.

Networks representing flow systems can be either closed or open loops. Open loop networks contain one or more boundary nodes that often represent the environment. In closed-loop networks the system pressure is determined by initial conditions. For such systems, if the flow is incompressible, the absolute value of pressure is indeterminate and MacroFlow will calculate the system pressure within an arbitrary constant. Similarly, the system enthalpy is determined by the environmental enthalpy (temperature).

In networks that are connected to the ambient, at least one node is a boundary node with known pressure and enthalpy. The flow enters or leaves the system through the links connected to these boundary nodes, and as such, provide a relative basis from which pressure and enthalpy can be defined.

The discretization of the mass, momentum, and energy equations and their solution is similar to the SIMPLE[6] algorithm. The solution of the momentum equations is carried out by explicit linkwise updates, while the solution of the mass and enthalpy equations involve field solutions.

The MacroFlow network representation of the experimental setup is shown in Fig. 12. The plenum is constructed from a combination of fans, straight exhausts, tees, nozzles, and connections. The specific component's selected model either a physical part of the plenum, such as one of the holes, or a flow abstraction, such as the tees. The tee component is used

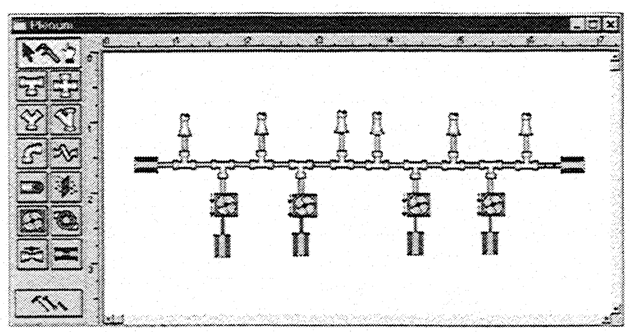

Fig. 12 Network representation used in MacroFlow to model the plenum.

to model the pressure loss incurred by the flow in turning 90 deg before exiting the plenum.

The network was defined by placing and connecting the components. Then, the specific characteristics for each component and connection were specified. This procedure involved selecting the component and connection icons and entering the required information into the associated component specific tab-dialogs. For example, for each of the connections that represent the plenum, the cross section, and length information were entered. Flow-driving components, such as the fans and the air supplies, required the specification of the head characteristics. These were entered in tabular form. After the component specification was completed, all of the tests were simulated. Results for the inhale/exhale pressure drops are presented in the next section.

Results and Discussion

To verify the exhaust fan performance, the pressure drop across the fan was plotted against the published values as shown in Fig. 13. Within the measurement accuracy (±5%), the data agree well with the published values.

Figure 14 shows the result of inhale and exhale pressure drop tests, i.e., series 1 and 2. The data are plotted against a curve of pressure drop as a function of the square of the volumetric flow rate, which is the theoretical prediction for the system. The experimental data at low flow rates (0–200 cfm) deviate from the predictions because the velocity measurements at this range were more difficult to determine as a result of the precision of the Alnor Velometer.

For a given opening flow cross-sectional area and a volumetric flow rate, the pressure drop is significantly lower during the inhale than the exhale mode of operation. Therefore, the exhale condition is more sensitive to pressure restrictions and flow rates and is the driving condition for sizing the bypass flow area and determining plenum pressure.

The results show that the pressure in the plenum was about 0.1-in. w.g. during the exhale of 400 cfm through an opening of ~0.6 ft^2 (~550 cm^2). This corresponds to an exhale condition of ~1200 scfm through an opening of ~1.8 ft^2 (~1700 cm^2) in the actual aircraft full plenum. Because the maximum exhale for operational scenarios are not expected to exceed 1200 cfm, a minimum of ~1.8 ft^2 appears adequate to meet the inhale/exhale requirements of the system.

As indicated before, a substantially larger flow area resulted in air being drawn into the plenum through the bypass holes closest to the inlet supply, and expelled through the holes toward the end of the plenum. The results indicate that a similar optimization and flow balancing is necessary on the as-built system.

For either the exhale or inhale modes of operation, the pressure within the plenum decreased as the number of operating fans increased. In addition, as the order or sequence in which the fans were turned off or on was changed, no noticeable effect in the system performance was observed. The maximum pressure drop within the plenum for the exhale case was below 0.5-in. w.g.

A comparison of the experimental measurements and the MacroFlow computational predictions is shown in Fig. 15. The agreement is excellent for exhale from 0.6 and 0.8 ft^2 openings. For larger openings during exhale, and all openings during the inhale, the predictions appear to underestimate the experimental measurements. The correlations incorporated into

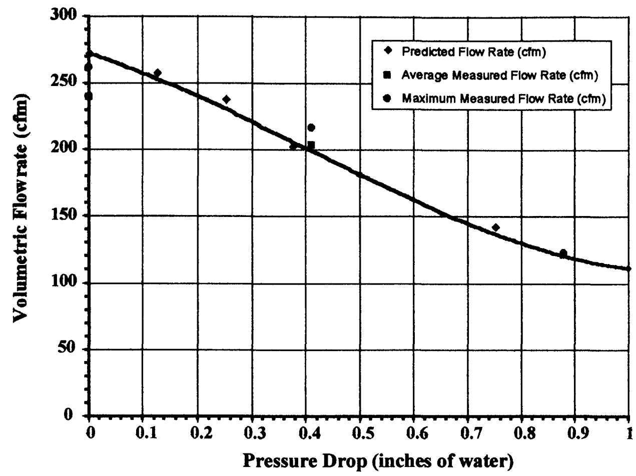

Fig. 13 Fantech FR150 characteristics.

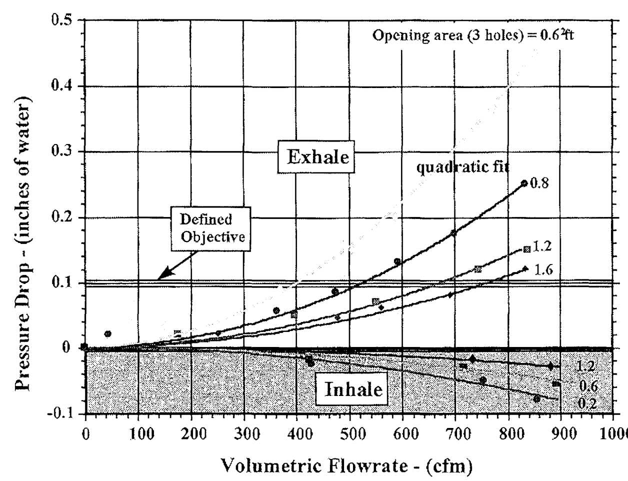

Fig. 14 Pressure drop in the plenum.

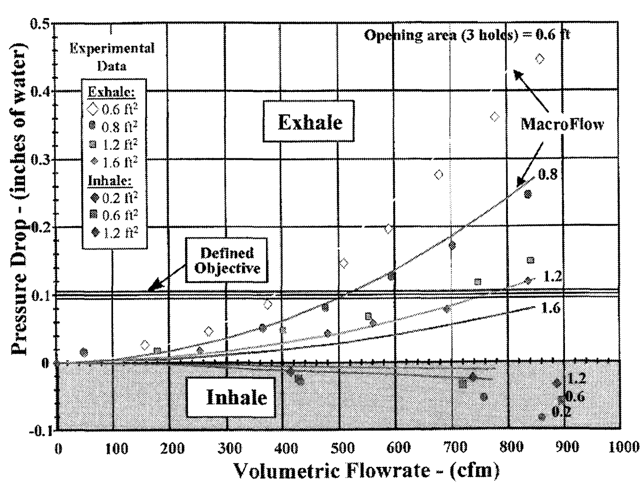

Fig. 15 Comparison of measured pressure drops with MacroFlow predictions.

MacroFlow do not account for any flow interactions between the exhausting flow from the holes. However, the exhausting flows do interact, especially in the case of when all six holes are open (the 1.2- and 1.6-ft^2 cases). An additional source of the apparent discrepancy between the experimental measurements and the MacroFlow predictions is perhaps caused by inaccuracies in measuring pressures below ±0.1-in. w.g., and low velocities. However, it is noted that in spite of these potential difficulties, the agreement is well within an acceptable engineering range.

It is of interest to note that if both the blower and the fans are kept on simultaneously, experimental measurements indicate that the flow rate through some of the holes will change direction. This effect was also captured by the MacroFlow prediction.

Conclusions

A common inhale/exhale opening in the ceiling plenum is feasible for the Challenger aircraft supplementary cooling system. Tested techniques have been developed and measurement devices have been selected to optimize the size of inhale/exhale opening in the aircraft.

A substantially larger opening area in the plenum is required for the exhale mode of operation than is needed for the inhale scenario. Flow deflectors near the openings are required to prevent bidirectional flow through the openings during the balanced flow mode of operation.

The highest pressure in the plenum was about 0.5-in. w.g. during exhale of 800 cfm through an opening of ~0.6 ft^2 (~550 cm^2). This corresponds to an unlikely exhale of 2100 scfm through an opening of ~1.6 ft^2 (~1500 cm^2) in the actual aircraft system, which is not predicted in any of the operational scenarios. The pressure in the plenum was about 0.1-in. w.g. during exhale of 400 cfm through an opening of ~0.6 ft^2 (~550 cm^2). This corresponding to an exhale of 1200 scfm through an opening of ~1.8 ft^2 (~1700 cm^2). The maximum exhale for all operational scenarios is not expected to exceed 1200 cfm.

Pressure drop along the plenum for all scenarios was negligible. Pressure in the plenum at the highest balanced flow rate (800 cfm) was about 0.01-in. w.g., which is well below the estimated pressure drop in the plenum (0.1–0.15 in. w.g.). Relative location of the fan(s) with respect to the openings and order of fan(s) sequencing did not have any measurable effect on either the flow pattern or the pressure drop.

Theoretical modeling and simulation of the flow were carried out using MacroFlow. The model predictions are in good agreement with the experimental measurements.

References

[1]Hashemi, A., Dyson, E., and Wong, H., "Cooling of Onboard High-Power Electronics Using Augmented Heat Rejection from Aircraft Skin," *Proceedings of the Symposium on Thermal Science and Engineering in Honor of Chancellor Chang-Lin Tien* (Berkeley, CA), 1995, pp. 465–474.

[2]Dyson, E., Hashemi, A., and Wong, H., "High-Power Electronics Heat Rejection from Aircraft Skin," *Journal of Enhanced Heat Transfer*, Vol. 3, No. 3, 1996, pp. 165–176.

[3]Hashemi, A., and Dyson, E., "Design of an Aircraft Skin Cooling System for Thermal Management of Onboard High Power Electronic Equipment," *Proceedings of the ASME 31st National Heat Transfer Conference*, HTD-Vol. 329, Vol. 7, American Society of Mechanical Engineers, New York, 1996, pp. 233–243.

[4]Hashemi, A., and Dyson, E., "Performance Characterization of High-Power Electronic Equipment Onboard an Aircraft," AIAA Paper 97-0596, Jan. 1997.

[5]*MacroFlow, User's Manual*, Innovative Research, Inc., Minneapolis, MN, 1995.

[6]Patankar, S. V., *Numerical Heat and Fluid Flow*, Hemisphere, New York, 1980.