NASA Technical Memorandum 103801

Simulation of Brush Insert for Leading-Edge-Passage Convective Heat Transfer

R.C. Hendricks
Lewis Research Center
Cleveland, Ohio

M.J. Braun and V. Canacci
University of Akron
Akron, Ohio

R.L. Mullen
Case Western Reserve University
Cleveland, Ohio

Prepared for the
Fourth International Symposium on Transport Phenomena in Heat and Mass Transfer
sponsored by the University of New South Wales
Sydney, Australia, July 14-18, 1991

NASA

SIMULATION OF BRUSH INSERT FOR LEADING-EDGE-PASSAGE CONVECTIVE HEAT TRANSFER

R.C. Hendricks
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135

M.J. Braun and V. Canacci
University of Akron
Akron, Ohio 44325

and

R.L. Mullen
Case Western Reserve University
Cleveland, Ohio 44106

ABSTRACT

Current and proposed high-speed aircraft experience high leading-edge heat transfer (to 160 MW/m^2; 100 Btu/in.^{2} sec) and surface temperatures to 1370 K (2000 °F). Without cooling, these surfaces could not survive. In one proposal the coolant hydrogen is circulated to the leading edge through a passage and returned to be consumed by the propulsion system. Simulated flow studies and visualizations have shown flow separation with a stagnation locus that isolates a zone of recirculation at the leading edge. Here the coolant heat transfer will be minimized where the free-stream flux is the highest; consequently the surface temperature increases.

This paper describes a novel method for mitigating the flow separation and the isolated recirculation zones by using a brush insert in the flow passage near the leading-edge zone. The resulting flow field sweeps the concave surface, implying significant increases in heat transfer.

1. INTRODUCTION

The high heat loads of high-speed flight and reentry vehicles require leading-edge cooling of protruding components. A sketch of a high-speed reentry body illustrates the nature and position of the shock relative to the leading-edge geometry (Fig. 1) with the highest fluxes occurring in the forward stagnation zone. Therefore, some aerodynamic shapes use the propellant for cooling prior to its ingestion by the combustor. The coolant passages of a body, a leading edge, or a cowl lip undergo abrupt changes in geometry, and coolant flow separation often occurs. Such evidence has been established in an earlier study by Braun et al. (1989). In this study

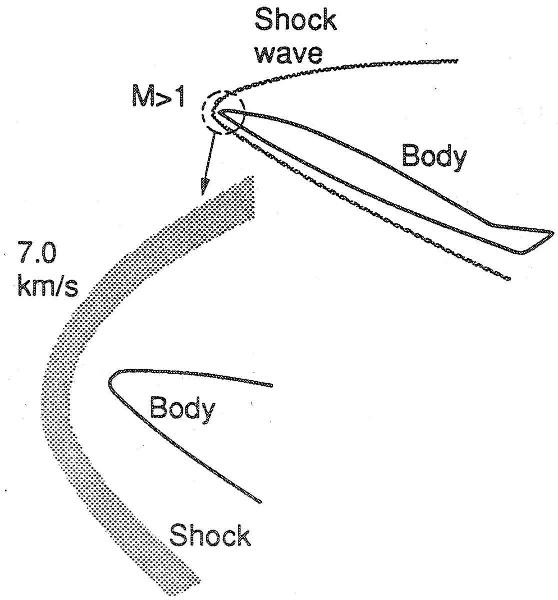


FIGURE 1. Sketch of reentry body and associated shock.

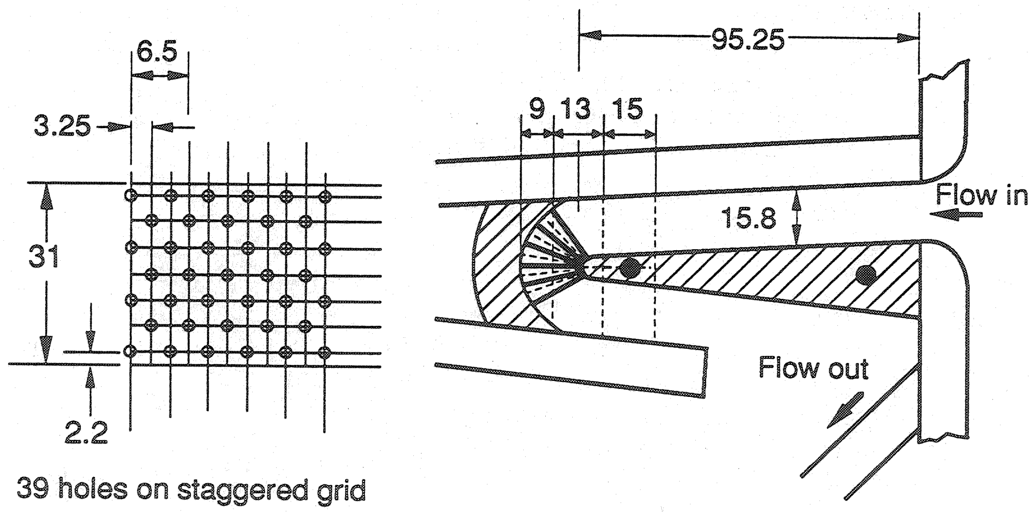
the same basic geometric configuration was retained, but the passage was modified by adding a cylindrical brush-like insert that had a sequence of staggered spokes. The flow in the modified and unmodified coolant passages was compared.

2. APPARATUS AND PROCEDURE

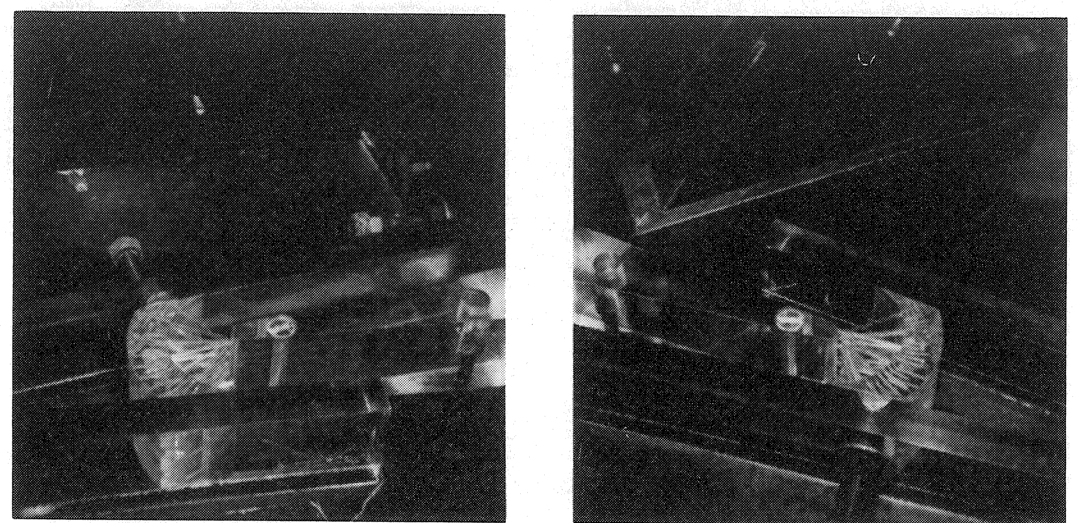
The basic geometry, water tunnel, and instrumentation are described by Braun et al. (1989) and a similar configuration was used for this study. The specific change was the addition of a brush-like structure within the flow passage (Fig. 2(a)). The brush consisted of 39 cylindrical fins (or spokes) 1 mm in diameter. The convex surface (inner surface) had a radius of 3.3 mm, and the concave surface (outer surface) was elliptical (the average radius was approximately 20.3 mm). The inlet flow area was smaller than the outlet flow area (Fig. 2(b)).

There were four layers (r-θ plane) with six spokes and three layers with five spokes. The spokes were staggered, and at the concave surface the spokes were equally spaced at 6.5 mm (Fig. 2(b)). The tunnel height was less than 32 mm, and a rubber spacer was placed on the undercarriage support structure to force the model squarely against the upper wall of the tunnel and thus deter leakage. The tunnel was a closed loop and the working fluid was distilled water. Although the tunnel was cleaned, some residual deposits and flow tracers remained within it.

The light source was a 1-W argon ion, green, continuous-wave laser. The illumination of the spokes or fins led to some light piping similar to fiber optics. The illuminated flow field was recorded on 1/2-in. video tape. Although somewhat obscure near the spoke (or bristle) and in the region near



(a) Sketch of insert geometry.



(b) Photographs of passage geometry.

FIGURE 2. Insert and passage geometry. (Dimensions are in millimeters.)

the convex interface, the flow patterns were sufficiently clear between the spokes to permit assessment and comparison with the results of Braun et al. (1989).

3. RESULTS AND DISCUSSION

For a fluid to turn in a curved channel, some form of secondary flow is required to enforce the balance of momentum. Usually, a large secondary vortex or pair of vortices form such that the central core moves from the convex to the concave surface and returns along the boundaries. The strength of these vortices is dependent on the curvature and aspect ratio of the duct.

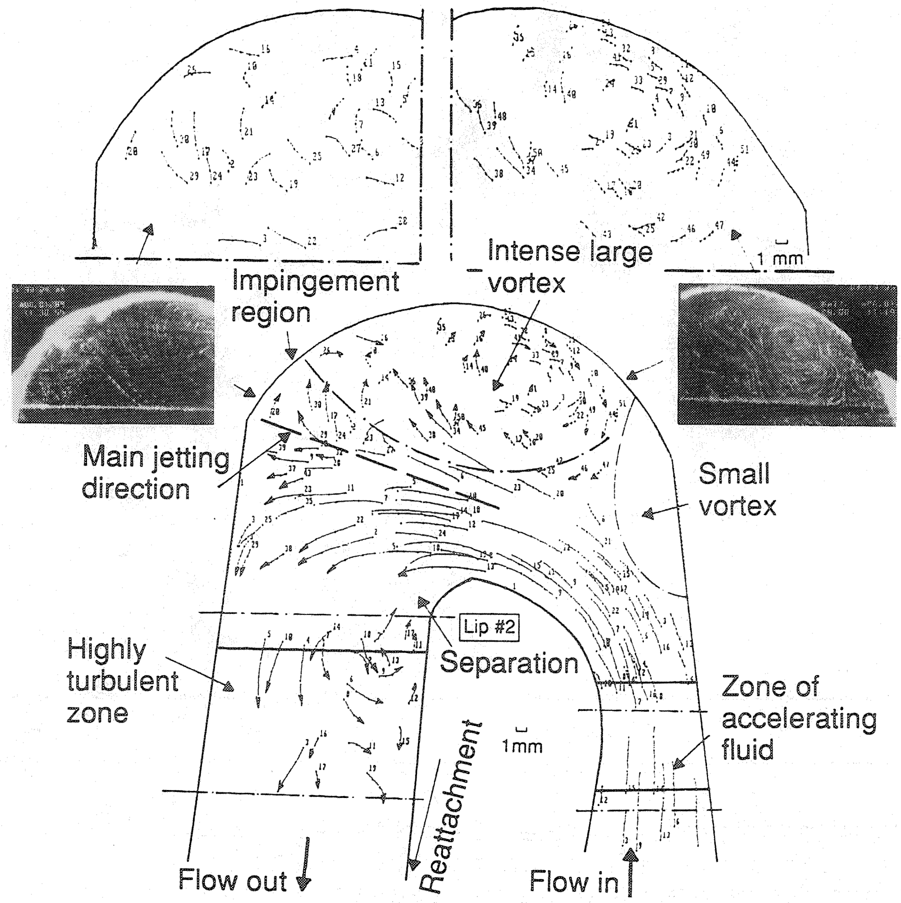
Flows in turning ducts all undergo changes in momentum with attendent losses. In most cases, secondary flow fields form for this purpose as viewed in the r-z plane, and separation is often encountered as viewed in the r-θ plane. The experiments of Braun et al. (1989) were viewed in the r-θ plane only (secondary flow in the r-z plane was not investigated). A stagnation point was found to form on the concave surface approximately 120° to 150° into the 180° turn of the flow field (Fig. 3) with a large vortex forming in the zone of maximum heat transfer. Also a separation zone formed immediately downstream of the small-radius turn at the convex surface. With reference to Fig. 1, the formation of such separation zones would tend to enhance heat transfer at the convex surface, but the stagnation and vortex zones at the concave surface could lead to either leading-edge failure or destruction of the vehicle.

The brush-like insert was designed to eliminate the stagnation zone at the concave interface while maintaining most of the separation zone at the convex interface. And the video representation of the flow field showed that the brush did just that (Figs. 3 and 4). The flow appears more as solid-body rotation yet highly three dimensional, and most of the separation zone appears downstream of the convex turn (see Figs. 3 and 4 and associated sketch of streamlines garnered from several different experiments).

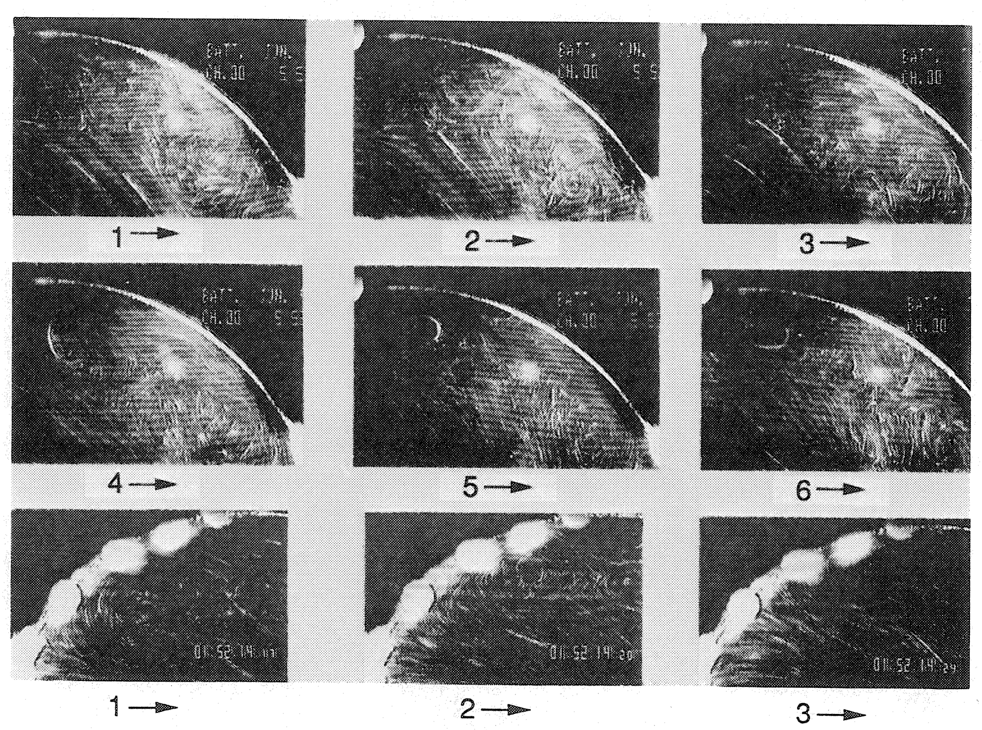
The brush-like insert did tend to complicate the geometry and to increase pressure drop. With the addition of the brush (Fig. 2) both the porosity and the flow area decreased about 5 and 10 percent, respectively, but the undesirable effects were eliminated. No attempt was made to optimize the configuration. The use of spokes or fins enhanced the heat transfer process through the addition of two mechanisms: (1) one is the formation of microvorticity zones in the vicinity of the spoke (or bristle), and (2) the other is represented by the action of the spoke as a natural fin (perhaps its effectiveness could be enhanced by use of beryllium copper). Structurally, the spokes made the coolant passage significantly stronger because the concave and convex surfaces were supported along the entire length of the leading edge and not just at local chords. A composite structure was formed.

The brush-like insert represents a simple solution to a complex problem.

Similar brush geometries could be used in many turning ducts with similar benefits. For example, flow field distortions and separation zones in the space shuttle main engine hot-gas turnaround duct have been assessed by several computational fluid dynamics calculations. The addition of a brush-like insert would tend to mitigate these effects by forcing the flow to turn in a more wheel-like manner. Another example is in two-phase, pseudo-two-phase, or other flows with large variations in fluid properties. Here the flow field is geometry specific. Heat transfer and pressure drop depend on the void fraction distribution and the single-phase parameters, the curvature, and the relation between the concave

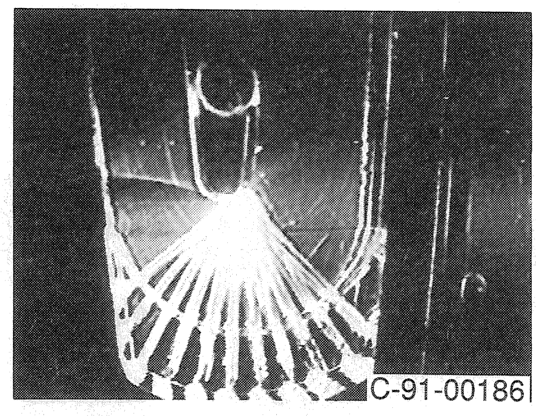


(a) Quantitative flow resolution.

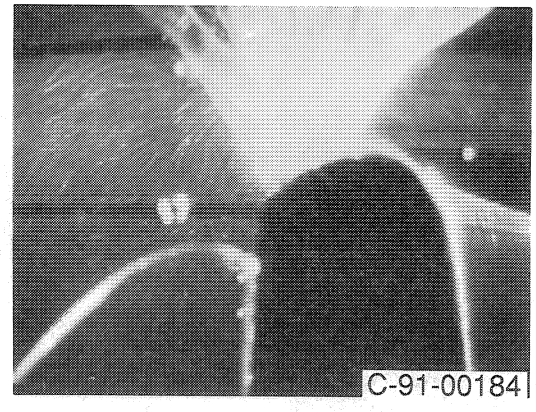


(b) Qualitative description of flow field.

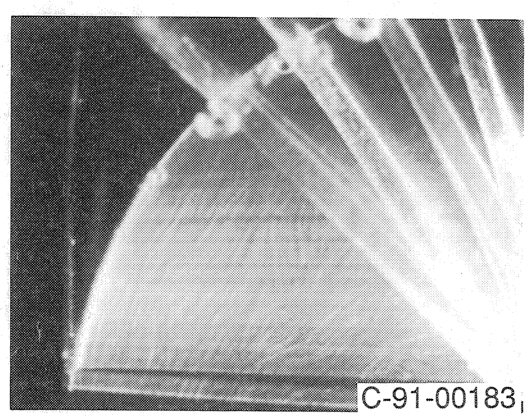
FIGURE 3. Flow patterns in standard turnaround passage, from Braun et al. (1989).



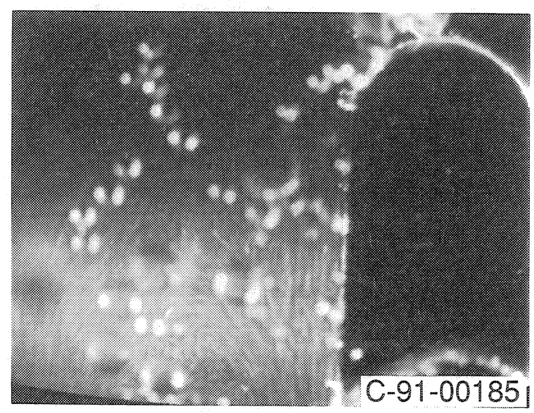
(a) Overview of geometry and flow patterns.



(c) Flow field near convex surface at 120°.



(b) Flow field at 120° downstream.



(d) Flow vortex downstream of separation zone.

FIGURE 4. Flow patterns in turnaround passage of Braun et al. (1989) with brush insert.

and convex surfaces (i.e., Dean number effects). For some configurations and flows the heavier fluid streamlines form closer to the convex interface. Heat transfer is enhanced at the convex surface and degraded at the concave surface, an undesirable effect for this application. The addition of a brush-like insert would also tend to induce fluid motion toward the concave interface, a desirable effect for this application.

4. CONCLUSIONS

A brush-like insert can be instrumental in eliminating stagnation zones in turnaround ducts and in forcing the flow toward solid-body rotation. The decreases in porosity and flow area are small, less than 10 percent, and the geometry becomes more complex. However, other benefits such as continuous structural support and the elimination of the undesirable flow field characteristics make the brush-like insert an ideal design modification.

APPENDIX - BRUSH INSERT DESCRIPTION

A brush with a staggered bristle pattern engenders sequences of smaller vortices that provide a greater reduction in momentum near the convex surface than at the concave surface (vortex coupling is desirable at the concave surface but undesirable at the convex surface). These vortex sequences couple to mitigate the secondary flow field, promote solid-body rotation, and provide smaller vortices to dissipate within the duct. In turn, these vortices enhance the heat transfer at the concave surface, where fluxes are the highest (Fig. 1), and the bristles enhance structural stiffness.

In order to force a uniform flow pattern in a curved duct, the flow resistance must vary inversely with the radius. This is achieved by the use of a brush-like insert, a set of spokes or bristles that are radial structural elements separating the inner and outer surfaces.

Geometry

The circumferential spacing between bristles of diameter d is $\Delta\theta$. The number of bristles in the circumferential direction becomes

$$N_\theta = \frac{\theta}{\Delta\theta} + 1 \tag{1}$$

where θ represents the bend angle, or the angle through which the brush acts, and cannot be greater than

$$(N_\theta - 1)[d_e + \varepsilon(r_i)] = r_i\theta \tag{2}$$

where d_e is the elliptical length at the interface and $\varepsilon(r_i) = \varepsilon_0$ is the spacing between bristles at the convex (inner) surface of length $r_i\theta$. Let the bristle spacing in the axial or z direction be Δz. With the first plane of bristles parallel to and at Δz from the surface in the r-θ plane, the number of bristle planes becomes

$$N_z = H/\Delta z \tag{3}$$

where H is the height of the cylindrical surface. For a staggered grid (Fig. 2) the total number of bristles becomes

$$\begin{aligned} N &= N_z(N_\theta - 1/2) \qquad (N_z \text{ even}) \\ N &= N_\theta N_z - (N_z - 1)/2 \qquad (N_z \text{ odd}) \end{aligned} \tag{4}$$

Porosity

The flow momentum losses will be related to porosity, which relates blockage as a function of radius.

$$P_{or} = V_{open}/V_{total} = 1 - (V_{solid}/V_{total}) \quad (5)$$

For this design, the tunnel is assumed to be two dimensional, and the spokes or bristles are considered circular (rods).

$$V_{open} = H\theta(r_0^2 - r_i^2)/2 - N\pi\, d^2(r - r_i)/4 \quad (6)$$

The global or average porosity over the channel can be calculated as

$$P_{or} = 1 - (N\pi\, d/2H\theta)\,[d/(r + r_i)] \quad (7a)$$

while local porosity becomes

$$P_{or} = 1 - (N_\theta \pi/2\theta)\,[d/(r + r_i)] \quad (\textit{in plane}) \quad (7b)$$
$$P_{or} = 1 \quad (\textit{between planes})$$

Flow Area

The flow area decreases for the staggered grid and can be approximated by using alternate bristle planes.

$$A = (r_0 - r_i)(H - dK); \quad K = (N_z + 1)/2 \quad (N_z\ \textit{odd}) \quad (8)$$
$$K = N_z/2 \quad (N_z\ \textit{even})$$

Friction Factor

The friction factor can be defined in terms of flow over a set of sequential cylinders (e.g., flows over pin-fin heat exchangers) or by flow in a porous medium where provisions are made for the radial variations in geometry.

One such porous medium relation is described for developed flows in linear ducts (Gunter and Shaw, 1945). For planar brush elements with planar end walls, the lateral spacing is given by

$$S_L = r\,\Delta\theta \quad (9)$$

and the transverse spacing is given by

$$(S_T/S_L)^2 = 1/4 + (\Delta z/r\,\Delta\theta)^2 \quad (10)$$

For nonplanar staggering it may be more appropriate to use the transverse spacing given by

$$S_T = 2\,\Delta z \quad (9a)$$

and the lateral spacing given by

$$(S_L/S_T)^2 = 1 + (r\,\Delta\theta/\Delta z) \tag{10a}$$

with the restriction (noted previously, Eq. (2))

$$r_i\,\Delta\theta > d_e + \varepsilon_0 \tag{11}$$

The equivalent hydraulic diameter becomes

$$D_v = 4V_{open}/A_{wetted} \tag{12}$$

The wetted surface can then be calculated from

$$A_{wetted} = (r_o + r_i)H\theta + (r_o^2 - r_i^2)\theta/2 - 2N\pi d^2/4 + N\pi\, d(r - r_i) \tag{13}$$

Using these parameters a friction factor can be calculated from Gunter and Shaw (1945).

$$f/2 = \rho\,\Delta P\phi/G^2 \tag{14}$$

where

$$\phi = \langle D_v/L\rangle(\mu/\mu_\infty)^{0.14}(S_T/\langle D_v\rangle)^{0.4}(S_T/S_L)^{0.6}$$

and

$$f/2 = 90/Re \qquad Re < 200 \qquad (laminar)$$
$$f/2 = 0.96/Re \qquad Re > 200 \qquad (turbulent)$$

and where

$$Re = G\langle D_v\rangle/\mu$$

and

$$L = (r + r_i)\theta/2$$

After rearranging Eq. (14) the laminar flow parameter becomes

$$90/D_v\phi = \Delta P/\nu G \tag{15}$$

and the turbulent flow parameter becomes

$$0.96/D_v^{0.145}\phi = (G/\mu)^{0.145}\rho\,\Delta P/G^2 \tag{16}$$

Illustration of Geometric Variation and Flow Parameters

Assume that the coolant passage is a set of concentric cylinders (not a requirement) similar to that of Fig. 2.

$r_o = 20\ mm,\quad d = 1\ mm,\quad r_i = 3.6\ mm,\quad N = 39,\quad N_z = 7,$
$N_\theta = 6,\quad H = 35\ mm,\quad Fluid = Water,$
$\theta = 2\pi/3,\quad No\ wall\ heating$

The variation in porosity is given as Fig. A-1, and the changes in the laminar and turbulent parameters (Eqs. (15) and (16)) are given in Fig. A-2.

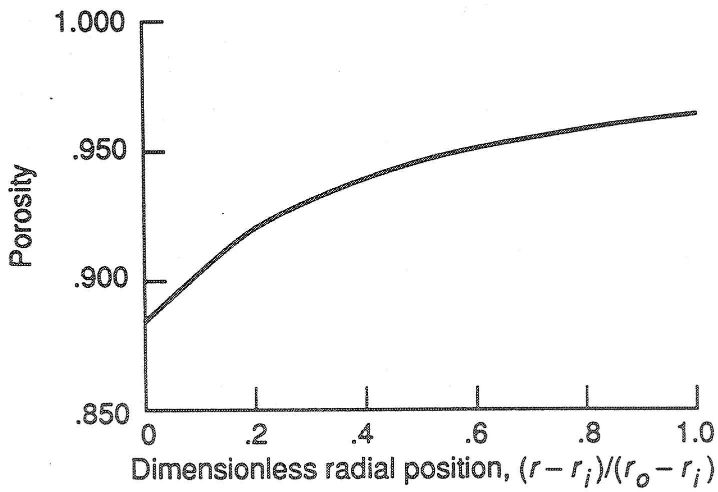


FIGURE A-1. Variation in porosity with radial position.

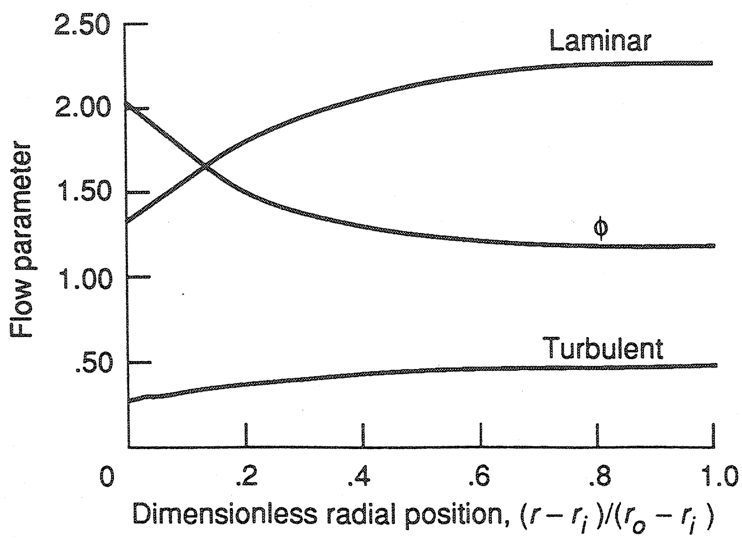


FIGURE A-2. Changes in laminar and turbulent parameters.

REFERENCES

Braun, M.J., Canacci, V.A., Russell, L., Hendricks, R.C. (1989) A Laser Based Computer Aided Non-Intrusive Technique for Full Field Flow Characterization in Macroscopic Curved Channels, in Flow Visualization - 1989, eds., B. Khalighi, M.J. Braun, and C.J. Freitas, ASME FED-Vol. 85, ASME, New York, pp. 15-22.

Gunter, A.Y., and Shaw, W.A., A General Correlation of Friction Factors for Various Types of Surfaces in Crossflow, ASME Trans., vol. 67, no. 8, pp. 643-660, Nov. 1945.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. AGENCY USE ONLY (*Leave blank*)

2. REPORT DATE
July 1991

3. REPORT TYPE AND DATES COVERED
Technical Memorandum

4. TITLE AND SUBTITLE
Simulation of Brush Insert for Leading-Edge-Passage Convective Heat Transfer

5. FUNDING NUMBERS
WU–505–62–52

6. AUTHOR(S)
R.C. Hendricks, M.J. Braun, V. Canacci, and R.L. Mullen

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135–3191

8. PERFORMING ORGANIZATION REPORT NUMBER
E–6093

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
National Aeronautics and Space Administration
Washington, DC 20546–0001

10. SPONSORING/MONITORING AGENCY REPORT NUMBER
NASA TM–103801

11. SUPPLEMENTARY NOTES
Prepared for the Fourth International Symposium on Transport Phenomena in Heat and Mass Transfer sponsored by the University of New South Wales, Sydney, Australia, July 14–18, 1991. R.C. Hendricks, NASA Lewis Research Center. M.J. Braun and V. Canacci, University of Akron, Akron, Ohio 44325. R.L. Mullen, Case Western Reserve University, Cleveland, Ohio 44106. Responsible person, R.C. Hendricks, 216–433–5912.

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Unclassified - Unlimited
Subject Category: 34

Available electronically at http://gltrs.grc.nasa.gov
This publication is available from the NASA Center for AeroSpace Information, 301–621–0390.

12b. DISTRIBUTION CODE

13. ABSTRACT ***(Maximum 200 words)***
Current and proposed high-speed aircraft experience high leading-edge heat transfer (to 160 MW/m^2; 100 Btu/in.^{2} sec) and surface temperatures to 1370 K (2000 °F). Without cooling, these surfaces could not survive. In one proposal the coolant hydrogen is circulated to the leading edge through a passage and returned to be consumed by the propulsion system. Simulated flow studies and visualizations have shown flow separation within the passage at the turnaround point of the center dividing passage with a stagnation locus that isolates a zone of recirculation at the most critical portion of the passage, namely the leading edge itself. This paper describes a novel method for mitigating the flow separation and the isolated recirculation zones by using a brush insert in the flow passage near the leading-edge zone, thus providing a significant increase in heat transfer.

14. SUBJECT TERMS
Heat transfer; Leading edges; Brushes

15. NUMBER OF PAGES
12

16. PRICE CODE
A03

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
Unclassified	Unclassified	Unclassified	