AMES GRANT
IN-34-CR
154833
13P.

NUMERICAL INVESTIGATIONS IN THREE-DIMENSIONAL INTERNAL FLOWS

SEMI-ANNUAL STATUS REPORT

1 OCTOBER 1987 THROUGH 31 MARCH 1988

Prepared for:

NASA-AMES RESEARCH CENTER

MOFFETT FIELD, CA 94035

UNDER NASA GRANT

NCC 2-507

(NASA-CR-183108) NUMERICAL INVESTIGATIONS IN THREE-DIMENSIONAL INTERNAL FLOWS Semiannual Status Report, 1 Oct. 1987 - 31 Mar. 1988 (Nevada Univ.) 13 p CSCL 20D
N88-29106
Unclas
G3/34 0154833

by:

WILLIAM C. ROSE

ENGINEERING RESEARCH AND DEVELOPMENT CENTER

UNIVERSITY OF NEVADA, RENO

RENO, NV 89557

PROGRESS REPORT

1 OCTOBER 1987 - 31 MARCH 1988

I. BACKGROUND

Renewed interest in simulation of high enthalpy flows brought about by interest in hypersonic vehicles has led to the refurbishment and renewal of ground based wind tunnel facilities. The National Aerospace Plane Program (NASP) is a program requiring the use of high enthalpy ground based facilities for many aspects of the testing vehicle and its propulsion system. Simulation of flows expected to be present at the entrance to the combustor of the propulsion system is of particular interest here. This portion of the propulsion system presents some of the more challenging problems in the simulation of the fluid flow due to the very high enthalpy values required for adequate simulation. The 100 megawatt arc heater at the NASA-Ames Research Center is one such facility that has the potential to simulate the required entering combustor conditions. This facility had been previously developed, but the nozzle simulating the entrance to a combustor in the range of Mach numbers of about 3 to 3.5 does not exist. The design of a nozzle for this facility for the NASP program brought about interest in the present work.

Because of the very high temperatures that the gas develops through the process of being heated by the electric arc, potential structural and thermal design problems arise due to the high heat transfer expected to occur between the gas and the solid walls of the nozzle. These can be particularly interesting to the design in the throat of the nozzle where active cooling systems will be required in order to prevent the nozzle from burning through, or at least eroding and changing its area ratio. Unfortunately, heat transfer rates that will allow adequate design of the active cooling system are difficult to come by and in the past have been generated as a result of experimental data and extrapolations to higher enthalpy conditions. The laminar heat transfer rates may be adequately calculated as long as separation within the converging-diverging passage near the vicinity of the throat is not allowed to occur. Similarly, correlations are available for adequate estimates for the

turbulent heat transfer rates as a result of the work by Bartz (Reference 1). The latter is a correlation of experimental data for the case where the boundary layer entering the nozzle throat is turbulent. The range of Reynolds numbers investigated previously is not high enough to include areas that are of current interest in high enthalpy ground based facilities, and in particular are about an order of magnitude below those expected to occur in the throat of the proposed 100 megawatt arc heater nozzle. For most practical applications, the boundary layer entering the nozzle throat is expected to be turbulent due to the length of run along the solid walls upstream of the nozzle in conjunction with the violent flow existing within the arc heating chamber itself. Although the Reynolds number for the 100 megawatt arc heater nozzle is higher than those available through correlation functions, this does not represent a significant limitation to the use of the Bartz correlations because, with increasing Reynolds number, the expected change in heat transfer coefficient (Stanton number) is not large. This behavior is typical of both skin friction and heat transfer coefficients for most turbulent flow situations at very large Reynolds numbers.

In another area of interest, the numerical simulation of fluid flows in numerous physical situations is popular and may be able to shed light on the current problem of nozzle design for the 100 megawatt arc heater nozzle. The numerical simulations of properties of the flow very near the wall, such as heat transfer and skin friction, are problematic in that the numerical grids upon which solutions to various applicable modeling equations are solved may not be fine enough to allow sufficiently detailed data concerning the near wall behavior to be resolved, even in an engineering sense. The present study investigates the numerical simulation of flows within the proposed nozzle and attempts to calculate the heat transfer rates that could be expected to occur within the throat region and presents these values for use in the design of the active cooling system.

II. INTRODUCTION

The difficulty of dealing with the calculation of near wall properties has recently been addressed in numerous works by Viegas, Rubesin and Horstman (References 2, 3 and 4). These authors have addressed the question of providing detailed near-wall resolution in modern numerical simulations through the use of empirical or semi-empirical information concerning the nature of the turbulent boundary layer near a surface under the effects of various pressure gradients. Their work has been applied in both two and three dimensional numerical simulations.

Previous numerical studies have been carried out for the flow expected to occur in two nozzle concepts for the 100 megawatt arc heater. These results, by Rose, Perkins and Serafini, were published at the Fourth NASP Symposium in Reference 5. These calculations dealt primarily with the nature of the outflow from the nozzle as effected by the shape of the throat and no particular attention was paid to the resolution of the near wall properties. Results from the study of Reference 5 indicated that a nozzle with a slit throat, as opposed to a nozzle with a square throat, produced a more uniform outflow condition, and thus was tentatively chosen to pursue in the design of the nozzle. These calculations were carried out with the full Navier-Stokes three-dimensional code described in Reference 6. The Kumar code was used because of the potential for small regions of separating and reattaching flow to exist within the assumed sudden contraction just upstream of the throat of the nozzle. The solutions shown in Reference 5 indicated that a small region of separation was present just upstream of the throat, however, with the grid spacing that could be used to allow the solution for the three dimensional flow fields to be obtained within a reasonable amount of time under the constraints of the explicit time accurate algorithm used in the method of Reference 6, it was not possible to even roughly estimate the throat heat transfer rates. Historically, it is known that regions of reattaching flow can produce very high heat transfer rates, and thus the present investigation into what heat transfer rates might exist within the throat region of the nozzle was undertaken. In Reference 5, it was shown that the flow throughout most of the nozzle remained two dimensional, that is the sidewall effects of the slit nozzle were negligible except just at the sidewalls themselves. This finding led to the conclusion relative to the present study that the effects of heat

transfer in the nozzle throat over most of the nozzle could be investigated with the use of two-dimensional equations simulating the observed nominally two-dimensional flow field. The two-dimensional Navier-Stokes code described in Reference 7 was used in this study. Investigations into the effect of the throat heat transfer rate using a combination of the full Navier-Stokes equations and the semi-empirical wall function approach described in References 2 - 4 were carried out during the present investigation and the remainder of the report discusses the findings relative to the heat transfer rate throughout the slit nozzle for the 100 megawatt arc heater.

III. RESULTS AND DISCUSSION

Initially, the two-dimensional Navier-Stokes code was used to investigate the flow within the nominally two-dimensional plane of the proposed 100 megawatt slit nozzle. Results from this study, carried out on a grid similar to that used for the full three dimensional calculations in Reference 5, showed that the flow away from the walls could be adequately modeled with the two dimensional code. This conclusion was determined from the general agreement between the calculated boundary layer thicknesses and exit Mach number distributions. Once the use of the two dimensional code had been demonstrated to be valid at least for most of the flow expected to exist throughout the three dimensional nozzle, a mesh refinement study was initiated to determine the effect of calculated surface heat transfer rates taken directly from the Navier-Stokes code. The results from this mesh refinement investigation are shown graphically in Figure 1. This figure indicates that as the mesh is refined, that is, as the distance from the wall to the first point in the numerical simulation grid is decreased, the calculated heat transfer rate increases. Nearly an order-of-magnitude variation in the calculated heat flux values occurs in various portions of the nozzle. The smallest of the grids (Beta = 1.001) placed the point near the wall within the laminar sublayer of the turbulent boundary layer. This was determined from examining values of the turbulent wall boundary layer parameter y^+. Thus one would expect the value of the heat transfer determined from this calculation to be quite close to the correct value. Unfortunately, the time required to carry this calculation out is prohibitive in terms of parametric investigations useful for engineering design procedures because the time step must be small for the small cell size. Thus, some technique for determining the near wall behavior of the turbulent boundary layer in the throat region is required that can be affected on a relative coarse numerical mesh.

The turbulent-boundary-layer wall-function approach described in References 2 - 4 was implemented for the algebraic turbulence model considered in present study. This implementation is in fact simpler than those described in the cited references for the wall function work because of

the algebraic turbulence model being used here rather than a full multiple equation turbulence model. A brief description of the use of these wall functions is given here for completeness. Equation 1 shows a law-of-the-wall equation for where constants C_1 and k are to be determined through empirical relationships. For mild pressure gradient flows and flows undergoing accelerations, these constants are well understood and can be given by Equation 2.

$$u^+_c = 1/k\ (\ln y^+) + C_1 \qquad (1)$$

$$u^+_c = 2.5 \ln y^+ + 5.1 \qquad (2)$$

Equation 2 describes the variation of the velocity away from the wall in the fully turbulent region of a turbulent boundary layer once the very near wall laminar sublayer and a brief transition distance between the laminar sublayer and the fully turbulent region has been passed. In the present code, no attempt was made to use a single equation describing the behavior all the way from the wall into the turbulent region. In general, beyond the turbulent region a wake exists whose size depends on the nature of the pressure gradient being encountered by the turbulent boundary layer. For highly accelerated flows, such as those encountered in the current nozzle, there is essentially no wake involved, and considerations of the nature of the wake portion of the turbulent boundary layer are neglected throughout the remainder of the present discussion. The crux of the implementation of this wall-function method is similar to a procedure used by experimentalists in fitting experimental data to an assumed mathematical function and deducing the wall shear stress or wall heat transfer values. In the present study, Equation 2 was used and values of the heat transfer rate were determined from points within the mesh at values of y^+ between 100 and 200. The remainder of the use of Equation 2 follows Reference 3, equations 10-34. This allows a very coarse mesh to be used (as opposed to the very fine mesh used to resolve a laminar sublayer) and accurate near-wall parameters to be obtained. Heat transfer results obtained in comparison with the very fine mesh of Figure 1 are shown for the wall function implementation on a coarse mesh in Figure 2. As can be seen, the wall function technique is very effective at producing a reasonable engineering result at a

much lower computation time. These results have been shown for the two dimensional code applied to the proposed geometry for the slit nozzle for a case where the total temperature is 4000°R and the walls are assumed to be held at 1000°R. The total pressure was 41 atm. As noted previously, separation exists in this throat because of the sudden contraction and maximum heating rates are as high as 12 MW per square meter. These conditions correspond to those in the Johns Hopkins University, APL, high enthalpy facility. Two other conditions were studied for the Ames 100 megawatt arc heater. They are: Ames #1; T_t = 6500°R, P_t = 53 atm. and Ames #2; T_t = 9180°R, P_t = 23.8 atm. All of the calculations assume that gamma is frozen at its stagnation value and the wall temperature remains at 1000°R. The solutions presented for these three cases were carried out on very coarse grids that allowed a speed-up of a factor of 100 from that case in which laminar sublayer resolution was obtained.

Figure 3 shows the final calculated heat transfer rates in the throat the slit nozzle with a new smooth contraction section for which no separation occurs. Values of up to about 6 MW per square meter can be expected to occur in the throat region itself for the Ames #2 condition. This calculated value may be compared with extrapolations of the heat transfer from Reference 1. The coefficient should have a value of about 0.0007. When multiplied by the mass flow and wall-to-total enthalpy difference, the dimensionalized heat transfer is about 5.0 MW per square meter near the throat. This use of a numerical simulation to determine an engineering number for design represents some of the first useful implementations of computational fluid dynamics applied to the NASP program. The active cooling system should be designed to be able to take out the indicated 6 MW per square meter expected to be delivered in the throat region. These values are within the range of heat fluxes that can be adequately cooled without excessive thermal gradients existing in the nozzle throat material (such as, for example, copper).

IV. CONCLUSIONS

An investigation into the use of computational fluid dynamics (CFD) has been carried out to examine the expected heat transfer rates that will occur within the NASA-Ames 100 megawatt arc heater nozzle. This nozzle has been tentatively designed and identified to provide research for a directly connected combustion experiment specifically related to the NASP aircraft, and is expected to simulate the flow field entering the combustor section. It was found that extremely fine grids, that is very small mesh spacing near the wall, are required to accurately model the heat transfer process and, in fact, must contain a point within the laminar sublayer if results are to be taken directly from a numerical simulation code. In the present study, an alternative to this very fine mesh and its attendant increase in computational time was invoked and is based on a "wall-function" method. It was shown that solutions could be obtained that give accurate indications of surface heat transfer rate throughout the nozzle in approximately 1/100 of the computer time required to do the simulation directly without the use of the wall-function implementation. Finally, a maximum heating value in the throat region of the proposed slit nozzle for the 100 megawatt arc heater was shown to be approximately 6 MW per square meter.

V. REFERENCES

1. Bartz, D.R.: Turbulent Boundary-Layer Heat Transfer from Rapidly Accelerating Flow of Rocket Combustion Gases and of Heater Air. Advances in Heat Transfer, Vol. 2, 1965.

2. Viegas, J.R. and Rubesin, M.W.: Wall-Function Boundary Conditions in the Solution of the Navier-Stokes Equations for Complex Compressible Flows. AIAA Paper 83-1694, July 1983.

3. Viegas, J.R., Rubesin, M.W., and Horstman, C.C.: On the Use of Wall functions as Boundary Conditions for Two-Dimensional, Separated Compressible Flows. AIAA Paper 85-0180, Jan. 1985.

4. Rubesin, M.W. and Viegas, J.R.: A Critical Examination of the Use of Wall Functions as Boundary Conditions in Aerodynamic Calculations. Paper presented at the Third Symposium on Numerical and Physical Aspects of Aerodynamic Flows, 20-24 Jan. 1985, California State University, Long Beach.

5. Rose, W.C., Perkins, E.W., and Serafini, D.B.: Navier-Stokes Analysis of Proposed Nozzles for the Ames 100 MW Heater. Paper No. 26 presented at the Fourth National Aero-Space Plane Technology Symposium, 17-19 Feb. 1988.

6. Kumar, A.: Numerical Simulation of Flow Through Scramjet Inlets Using a Three-Dimensional Navier-Stokes Code. AIAA 18th Fluid Dynamics Conference, July 1985.

7. Kumar, A.: Numerical Analysis of the Scramjet-Inlet Flow Field by Using Two-Dimensional Navier-Stokes Equations. NASA Tech. Paper No. 1940, Dec. 1981.

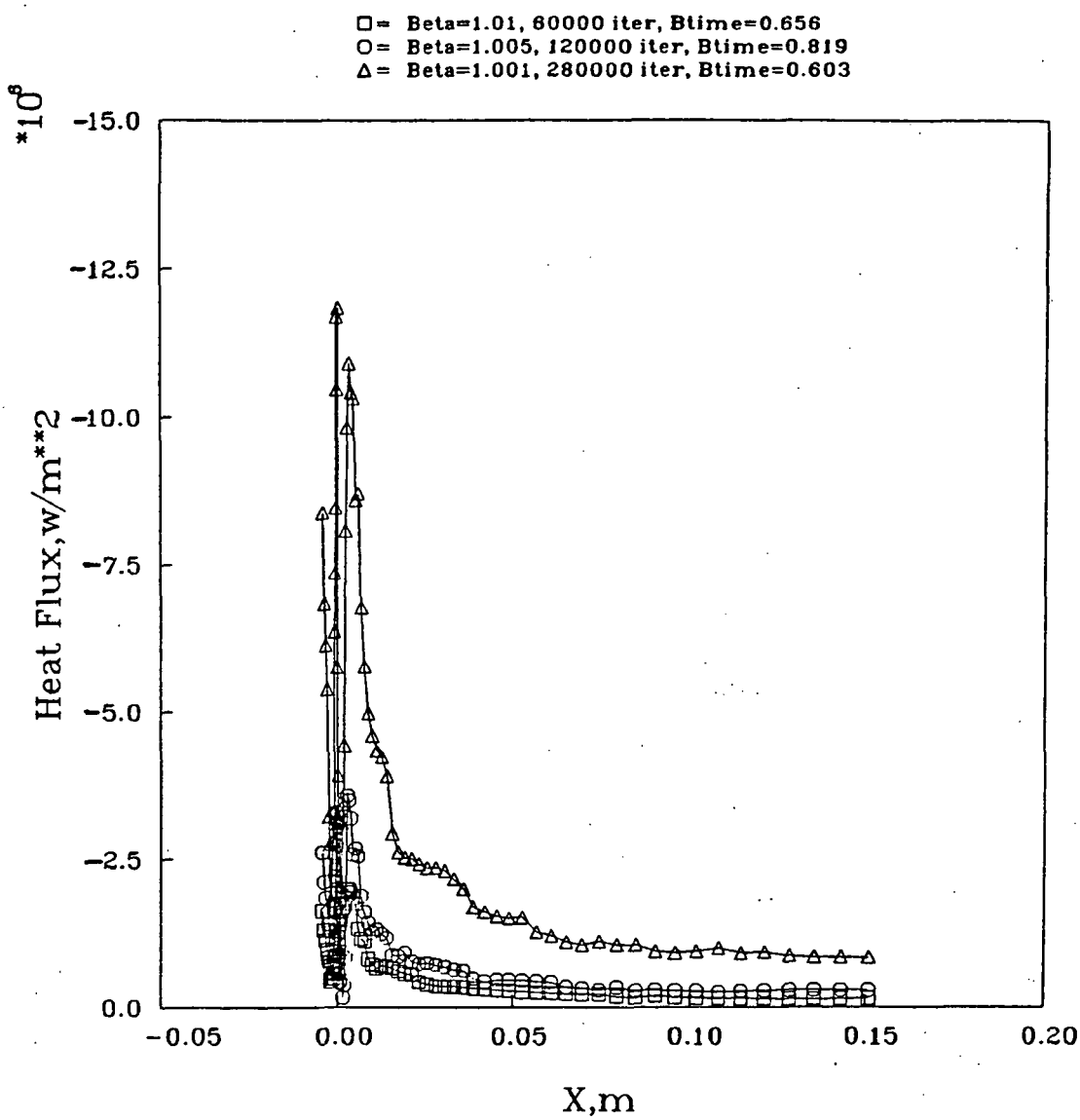

FIGURE 1. Heat transfer rates taken directly from the N-S code for various mesh spacing parameters for the APL slit nozzle with sudden contraction. (Beta = 1.001 is in laminar sublayer).

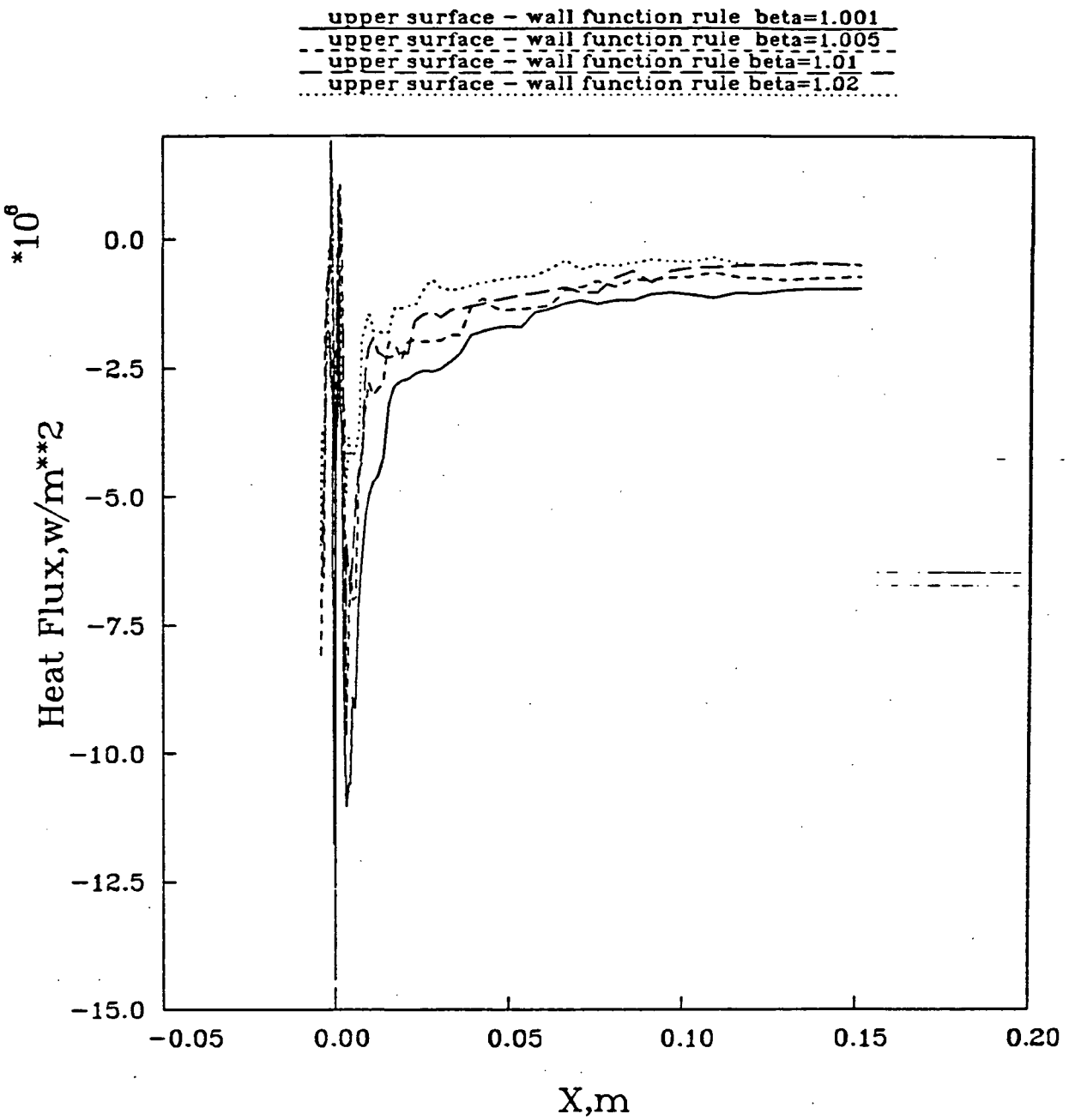

FIGURE 2. Heat transfer rates taken from the wall-function/N-S code for various mesh spacing parameters for the APL slit nozzle with sudden contraction (Beta = 1.001 is in laminar sublayer and wall function defaults to the direct code value).

ORIGINAL PAGE IS
OF POOR QUALITY

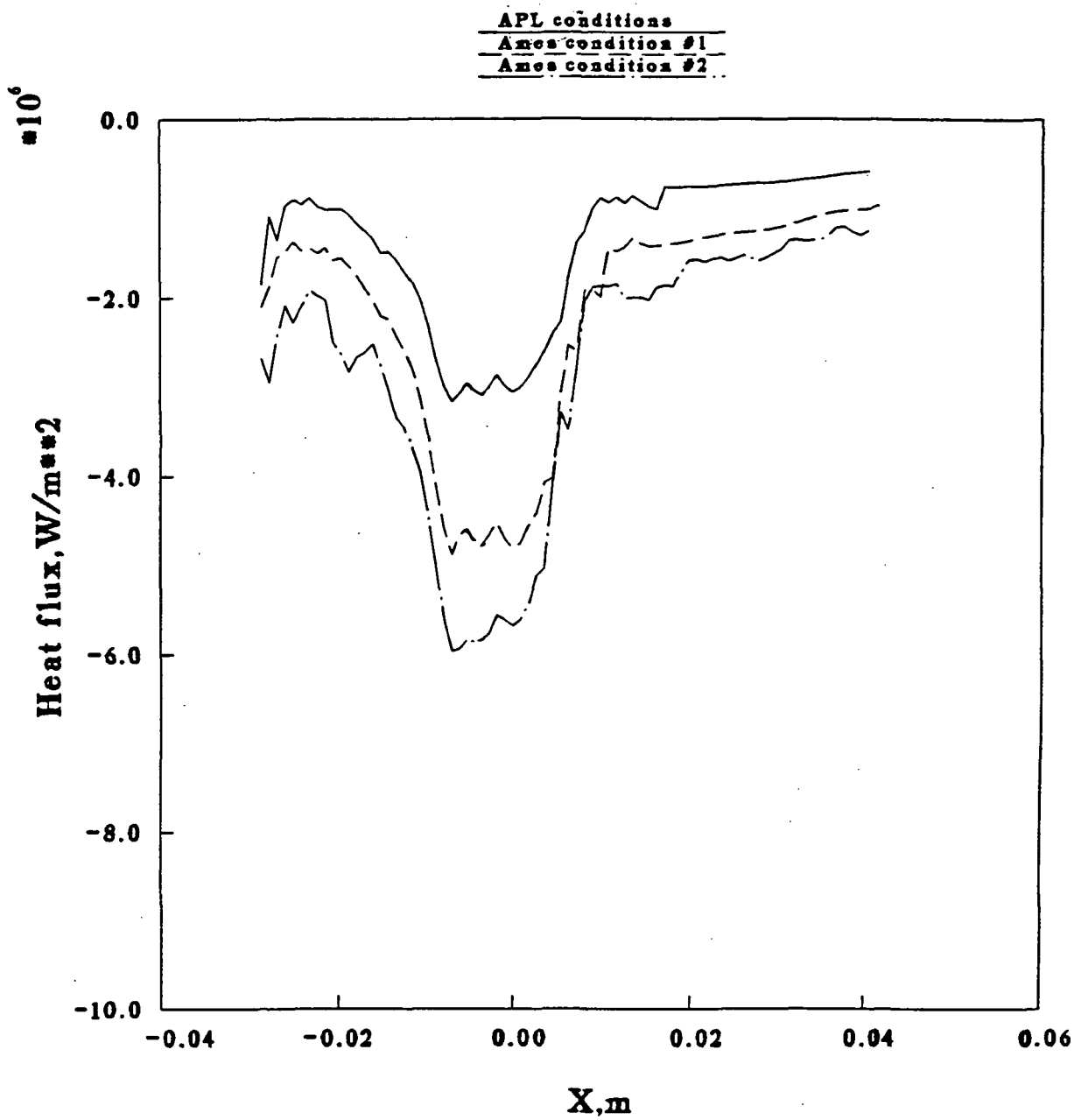

FIGURE 3. Heat transfer rates using wall-function/N-S code for the APL slit nozzle throat with new smooth contraction.