



### Approximate Solution for Convective Fins With Variable Thermal Conductivity<sup>1</sup>

A. Aziz.<sup>2</sup> The author compares in Fig. 3 the fin efficiency results based on the Galerkin method with those obtained using perturbation approach of the present discussor [1].<sup>3</sup> In [1] an error had crept in the derivation of equations for heat flux and fin efficiency. These were identified and corrected by Krane [2]. It seems that in the work under discussion the author used the incorrect fin efficiency equation of [1] for comparison appearing in Fig. 3. No wonder that he finds the perturbation results to be in significant error compared with the numerical results and his own approximate results based on Galerkin-type approach. If one uses the correct perturbation solution for fin efficiency [2] and compares the results as shown in Fig. 1, both the perturbation and the author's results are seen to be in good agreement

with the numerical results throughout the range of  $\epsilon$  from  $-0.6$  to  $+0.6$ . Furthermore, it must be emphasized that the perturbation solution is much simpler compared to the author's solution and that it has the additional merit of facilitating the evaluation of optimum fin dimensions. With the author's Galerkin type solution it is not possible to carry out the optimisation procedure.

#### Additional References

- 1 Aziz, A., and Enamul Huq, S. M., "Perturbation Solution for Convecting Fin With Variable Thermal Conductivity," JOURNAL OF HEAT TRANSFER, TRANS. ASME, Series C, Vol. 97, May 1975, pp. 300-301.
- 2 Krane, R. J., Discussion on a previously published paper by A. Aziz and S. M. Enamul Huq, JOURNAL OF HEAT TRANSFER, TRANS. ASME, Series C, Vol. 98, Nov. 1976, pp. 685-686.

#### Author's Closure

As stated in the paper under discussion, the comparison reported in Fig. 3 is referred to the fin efficiency equation given in [1]. Since this relationship has proved incorrect the comparison turns out to be misleading. In fact, the correct solution developed by Krane [2] shows a good agreement with the numerical results. In this connection it may be worthwhile to remark that there seems to be a slight discrepancy between the numerical results shown plotted in Fig. 3 and those reported by the discussor in Fig. 1. For instance, with  $N = 2$  and  $\epsilon = -0.6$  from the discussor's results one can estimate  $\eta = 0.393$  while the

<sup>1</sup> By A. Muzzio, published in the Nov. 1976 issue of the JOURNAL OF HEAT TRANSFER, TRANS. ASME, Series C, Vol. 98, pp. 680-682.  
<sup>2</sup> Assoc. Professor, Department of Mechanical Engineering, College of Engineering, Riyadh, Saudi Arabia.  
<sup>3</sup> Numbers in brackets designate Additional References at end of discussion.

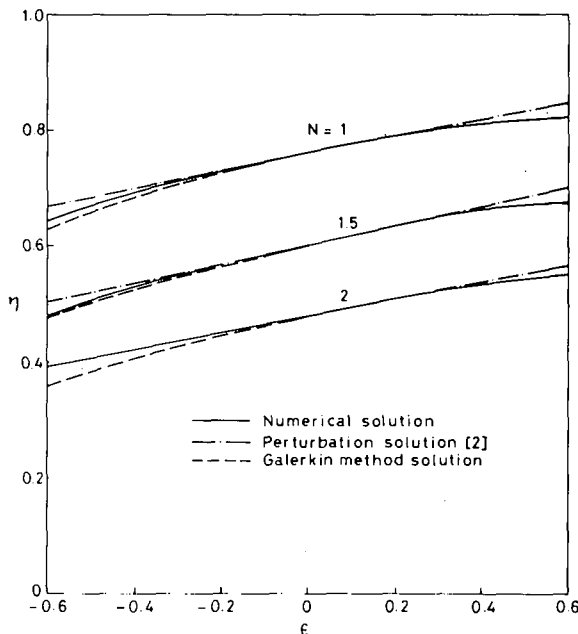


Fig. 1: Aziz

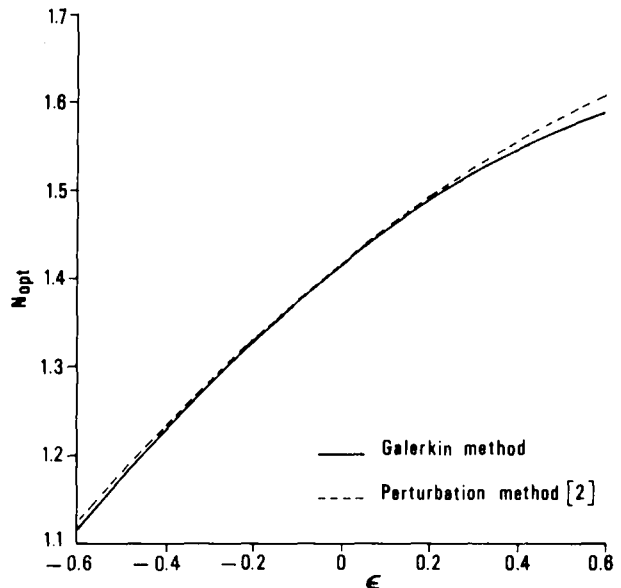


Fig. 1: Muzzio

author finds  $\eta = 0.378$ .

In any case, the Galerkin method solution seems to give a better accuracy, even though at the cost of a greater computation effort. Furthermore, it may be readily seen that, contrary to the discussor's claim, there is no difficulty in carrying out the optimization procedure.

The optimum values of  $N$  are obtained by solving the equation:

$$(\sinh 2N - 6N) \left( 1 - C \frac{\tanh^2 N}{1 + \tanh^2 N} \right)$$

## The Numerical Prediction of the Turbulent Flow and Heat Transfer in the Entrance Region of a Parallel Plate Duct<sup>1</sup>

T. Cebeci,<sup>2</sup> R. S. Hirsh,<sup>3</sup> and K. C. Chang.<sup>4</sup> The two previous calculations of entry flow in a duct [1, 2]<sup>5</sup> have both used rather complex turbulence models, i.e., a mixing-length model with twelve constants, and a  $k-\epsilon$  model which requires six constants. The present communication describes a much simpler approach to this problem based on the essential physics of the duct flow. As such it is applicable only to this particular flow, but a similar concept has been utilized for the calculation of shock wave/boundary-layer interactions [3].

The entry flow in a duct is shown schematically in Fig. 1. There are three identifiable regions in the flow field. In region I the flow behaves as an external boundary layer interacting with an inviscid flow; this is called the "displacement-interaction" region. By region III, the character of the flow is wholly described by internal concepts; this is the "fully developed" region. Region II is a transition zone from external to internal flow.

In region I, we use the well-tested Cebeci-Smith eddy-viscosity model [4] for the boundary-layer-like development. This is given by

$$\epsilon^I = \begin{cases} (0.4y)^2 [1 - \exp(-y/A)]^2 \left| \frac{\partial u}{\partial y} \right| \gamma_{tr} & (1a) \\ 0.0168 \int_0^\infty (u_e - u) dy \cdot \gamma_{tr} & (1b) \end{cases}$$

In a pipe, the flow in region III can be described by the classical mixing length model of Nikuradse [5]

$$\epsilon^{III} = \ell^2 \left| \frac{\partial u}{\partial y} \right| [1 - \exp(-y/A)]^2 \quad (2a)$$

$$\ell = r_0 \left[ 0.14 - 0.08 \left( 1 - \frac{r}{r_0} \right)^2 - 0.06 \left( 1 - \frac{r}{r_0} \right)^4 \right] \quad (2b)$$

where for axisymmetric flow for constant  $r_0$

$$r = r_0 - y$$

so that

$$1 - \frac{r}{r_0} = \frac{y}{r_0}$$

<sup>1</sup> By A. F. Emery and F. B. Gessner, published in the Nov. 1976 issue of the JOURNAL OF HEAT TRANSFER, TRANS. ASME, Series C, Vol. 98, pp. 594-600.

<sup>2</sup> Chief Aerodynamics Engineer, Research, Douglas Aircraft Co., Long Beach, Calif.

<sup>3</sup> Senior Engineer/Scientist, Douglas Aircraft Co., Long Beach, Calif.

<sup>4</sup> Research Associate, California State University at Long Beach, Long Beach, Calif.

<sup>5</sup> Numbers in brackets designate Additional References at end of discussion.

$$+ 6N \frac{dC}{dN} \frac{\tanh^3 N \cosh^2 N}{1 + \tanh^2 N} + 12NC \frac{\tanh^2 N}{(1 + \tanh^2 N)^2} = 0$$

where:

$$\frac{dC}{dN} = \frac{\alpha'C^2 + \beta'C + \gamma'}{2\alpha C + \beta}$$

The primes denote differentiation with respect to  $N$ . While the details of the calculation are not reported here, the results are shown plotted in Fig. 1 along with the perturbation solution [2].

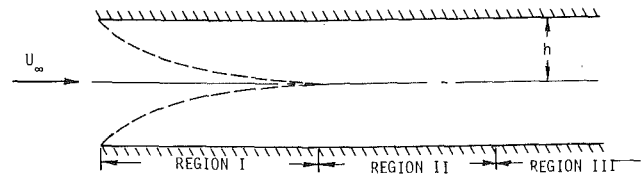


Fig. 1 Schematic representation of duct flow

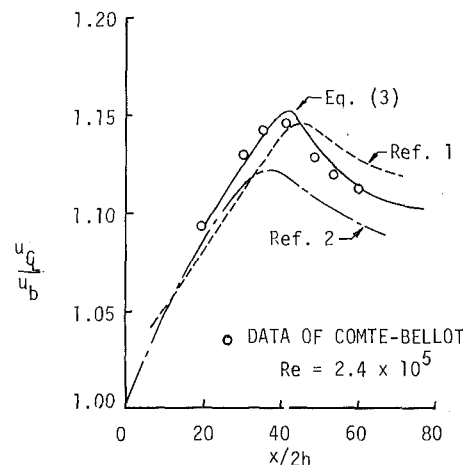


Fig. 2 Comparison of predictions for center-line velocity

To use this for two-dimensional duct flows we simply set  $r_0 = h$  to get

$$1 - \frac{r}{r_0} = \frac{y}{h} \quad (2c)$$

Either one of these can be used separately to calculate the developing duct flow using the numerical procedure described below, however, the overall agreement is poor. The solution was to model the entire flow situation based on our knowledge of the physics. A composite eddy viscosity is prescribed which changes smoothly from an external model ( $\epsilon^I$ ) to an internal model ( $\epsilon^{III}$ ) as follows

$$\epsilon = \epsilon^I + (\epsilon^{III} - \epsilon^I) \left[ 1 - \exp - \left( \frac{x - x_0}{20h} \right) \right] \quad (3)$$

where  $x_0$  is the length down the pipe where the two "external" boundary layers merge, and the denominator giving the relaxation scale was prescribed by divine intervention.

Using this model, the "nonlinear eigenvalue" procedure of Cebeci and Keller for laminar flows [6] was used to calculate the developing duct flow. For further details and heat transfer results see reference [7]. Fig. 2 shows the experimental results of Compte-Bellot for center-line velocity decay, along with the predictions of references [1, 2], and equation (3). The composite model provides strikingly improved results. The same comparison of predicted results for the displace-