An NPARC Turbulence Module
With Wall Functions

J. Zhu and T.-H. Shih
Institute for Computational Mechanics in Propulsion
and Center for Modeling of Turbulence and Transition
Cleveland, Ohio

August 1997

Prepared for
Lewis Research Center
Under Cooperative Agreement NCC3–370
AN NPARC TURBULENCE MODULE WITH WALL FUNCTIONS

J. Zhu * and T.-H. Shih
Center for Modeling of Turbulence and Transition
ICOMP, OAI, NASA Lewis Research Center
Cleveland, Ohio 44135

Abstract

The turbulence module recently developed for the NPARC code has been extended to include wall functions. The Van Driest transformation is used so that the wall functions can be applied to both incompressible and compressible flows. The module is equipped with three two-equation $k-\varepsilon$ turbulence models: Chien, Shih-Lumley and CMOTT models. Details of the wall functions as well as their numerical implementation are reported. It is shown that the inappropriate artificial viscosity in the near-wall region has a big influence on the solution of the wall function approach. A simple way to eliminate this influence is proposed, which gives satisfactory results during the code validation. The module can be easily linked to the NPARC code for practical applications.

1. Introduction

A turbulence module (Zhu and Shih, 1995) has been developed for the NPARC code. The module is written in a self-contained manner so that the user can use any turbulence model built in the module without concern as to how it is implemented and solved. The input to the module is the mean flow variables, boundary and geometric information which are to be provided by the NPARC code. The output of the module is the turbulent eddy-viscosity $\mu_t$ and/or relevant turbulent source terms which are needed for the mean flow calculation. The interaction between the NPARC code and the turbulence module will give the final turbulent flow solution. The turbulence module is conceived as a vehicle to facilitate the technology transfer from the model development to the model application. With the module, any development in the turbulence modeling can be quickly and directly validated by and applied to calculations of flows of engineering interests.

So far, three low Reynolds number two-equation turbulence models have been built into the module: Chien (1982), Shih and Lumley (1993) and CMOTT realizable models (Yang et al., 1995; Zhu and Shih, 1995). Unlike the Chien's model, both the Shih-Lumley and CMOTT models do not involve the dimensionless wall distance $y^+$, an advantage for separated flow calculations. The module with these turbulence models has been applied to a number of flows including flows over a flat plate, in an ejector nozzle, in a transonic diffuser, and a boat-tail nozzle flow (Yang et al., 1995). For all the flow cases tested so far, it has been found that both the Shih-Lumley and CMOTT models produce improved or similar predictions compared with the Chien model. The CMOTT model with variable $C_\mu$ turns out to be more computationally robust than the other two. It was able to give numerical solutions in cases where the models with constant $C_\mu$ suffered from numerical instability.

The major problems or difficulties associated with the low Reynolds number turbulence models are: 1) They require very fine grid spacing in near-wall regions, thus increasing considerably computational burden, especially in three-dimensional cases. Moreover, the highly stretched nature of mesh distribution may have an adverse impact on numerical stability. 2) Most of models are not of tensorial invariant form, that is, they contain a distance parameter normal to the wall. The wall-distance dependency causes inconvenience for model applications in complicated geometries. Currently, great effort is being given in the area of near-wall turbulence modeling to remove this dependency, but no satisfactory result has been obtained yet. 3) Most of the low Reynolds number turbulence models were finely tuned against attached flows, which is, of course, not sufficient to guarantee their good performance for separated flows.

An alternative is to use the high Reynolds number turbulence models. Here, the governing equations are integrated to a point far outside the vis-
von sublayer rather than down to the wall, and the near-wall region is bridged over with the wall functions. Although in theory the wall function approach is only valid for certain attached flows with no pressure gradient and mass transfer, it has been applied in practice to many separated flows with varying degree of success. For those flows where maximum shear stresses occur far away from the wall, the near-wall turbulence modeling is not critical for overall flow simulations. In these cases, use of the wall functions has a very beneficial effect on the stability and economy of computations. Although the principle argument for originally adopting the wall function approach (economy of grid points) has been weakened as larger and faster computers have become available, it will still find its applications in predicting complex flows, especially for large scale engineering problems.

In the present work, we extend the turbulence module by including the wall functions. For incompressible flows, the universal law of the wall may be expressed as

\[ \frac{U}{U_r} = \frac{1}{\kappa} \ln \left( \frac{\rho U_r y}{\mu} \right) + C \]  

(1)

where \( \kappa = 0.41 \) and \( C = 5.2 \). The derivation of Equation (1) is based on the assumption that the shear stress in the region close to the wall is constant and equal to the wall shear stress. It has been shown (Viegas et al., 1985; Huang and Coakley, 1993) that the same form also exists for compressible flows with the velocity \( U \) being replaced by the Van Driest transformed velocity \( U_c \) (Van Driest, 1951). For the \( K\)-\( \varepsilon \) turbulence models, the convection and diffusion terms of their transport equations are negligible in the inertial sublayer so that local equilibrium prevails, which implies that the production of the turbulent kinetic energy \( K \) is equal to the dissipation rate \( \varepsilon \) of \( K \). The local equilibrium condition leads to two simple relations which can be used as boundary conditions for \( K \) and \( \varepsilon \) for both incompressible and compressible flows. The compressible wall functions have been successfully applied to both attached and separated flows under Mach number ranging from 0.1 to 10 (Huang and Coakley, 1993).

Although the wall functions look simple, their numerical implementation is not trivial. The main difficulty comes from the logarithmic law in which both \( U \) and \( U_r \) are unknown, and \( U_r \) cannot explicitly be solved for. It is prone to being numerically unstable if one uses Equation (1) and iteratively solves \( U_r \) to obtain the boundary conditions for the Navier-Stokes equations. In this work, we use an implicit procedure which directly incorporates Equation (1) into the Navier Stokes equations. In this way, there is no need to solve Equation (1) for \( U_r \) by sub-iteration. The implicit method turns out to be more stable than the explicit one. Another important issue is the artificial viscosity. Chitsomboon (1994) found that the artificial viscosity originally implemented in the NPARC code totally spoiled the solution of the wall functions. This was because the artificial viscosity became unrealistically large in the vicinity of walls due to very steep velocity gradients resulting from the coarseness of grid spacing as required by the wall function approach. He fixed up this problem by extrapolating velocities at the wall rather than using the physical values of no-slip velocities, when calculating the artificial viscosity. In the present work, we simply turn off the artificial viscosity in the near-wall region.

In what follows, we will present the details of the wall functions and their implementation, and demonstrate how to link the turbulence module to the NPARC code. Detailed applications will be reported in another paper.

2. Calculation Approach

The following presentation is only restricted to the numerical aspects related to the implementation of the wall functions. Refer to Cooper and Sirbaugh (1989 and 1990) about the details of the NPARC code and to Zhu and Shih (1995) about the details of the turbulence module.

2.1 Governing Equations

In the NPARC code, the following nondimensional, Reynolds-averaged Navier-Stokes equations are solved:

\[ \frac{\partial \tilde{Q}}{\partial t} + \frac{\partial \tilde{F}_1}{\partial \xi} + \frac{\partial \tilde{F}_2}{\partial \eta} = \frac{\partial \tilde{G}_1}{\partial \xi} + \frac{\partial \tilde{G}_2}{\partial \eta} \]  

(2)

where \( \tilde{Q} \), \( \tilde{F}_j \), and \( \tilde{G}_j \) (\( j = 1, 2 \)) are the conservation variable vector, inviscid flux vectors and viscous flux vectors, respectively. Because only the viscous flux vectors need to be modified with the use of the wall functions, their detailed forms are given below:

\[ \tilde{G}_1 = \begin{bmatrix} g_{11} \\ g_{12} \\ g_{13} \\ g_{14} \end{bmatrix}, \quad \tilde{G}_2 = \begin{bmatrix} g_{21} \\ g_{22} \\ g_{23} \\ g_{24} \end{bmatrix} \]  

(3)
\[ g_{11} = 0 \]

\[ g_{12} = \hat{g}_{11} = J^{-1}(\xi_{t} n_{1} + \xi_{y} n_{2}) \]  

\[ g_{13} = \hat{g}_{12} = J^{-1}(\xi_{t} n_{21} + \xi_{y} n_{22}) \]

\[ g_{14} = U\hat{g}_{11} + V\hat{g}_{12} - J^{-1}(\xi_{t} q_{1} + \xi_{y} q_{2}) \]  

\[ g_{21} = 0 \]

\[ g_{22} = \hat{g}_{21} = J^{-1}(\eta_{t} n_{1} + \eta_{y} n_{2}) \]  

\[ g_{23} = \hat{g}_{22} = J^{-1}(\eta_{t} n_{21} + \eta_{y} n_{22}) \]

\[ g_{24} = U\hat{g}_{21} + V\hat{g}_{22} - J^{-1}(\eta_{t} q_{1} + \eta_{y} q_{2}) \]  

\[ U_{c} = \sqrt{B \left[ \sin^{-1} \left( \frac{A + U}{D} \right) - \sin^{-1} \left( \frac{A}{D} \right) \right] } \quad (12) \]

where

\[ A = \left( \frac{q}{\gamma} \right)_{wall} \]

\[ B = \frac{2 T_{wall}}{(\gamma - 1) P_{r}} \]

\[ D = \sqrt{A^{2} + B} \]

In the near-wall region, with the convection neglected the energy equation can be reduced to give an expression for the total heat flux

\[ q = q_{wall} + U \tau \]  

and with the local equilibrium assumption (Launer and Spalding, 1974), the turbulent kinetic energy \( K \) and its dissipation rate \( \epsilon \) can be calculated by

\[ K = \frac{\tau_{wall}/\rho}{0.3} \]  

\[ \epsilon = \frac{(\tau_{wall}/\rho)^{3/2}}{\kappa y} \]  

In the above expressions, the subscript wall refers to the value of the corresponding function at the wall. Equations (10) and (14) - (16) form the wall functions which are used to bridge over the first grid point and the solid wall.

2.3 Numerical Implementation

From Equations (10) and (11), the near-wall shear stress can be written as

\[ \tau = \tau_{wall} = \rho_{wall} U_{r} \frac{U_{c}}{U_{r}} = \frac{\mu_{t} U}{R_{e} y} \]  

where \( \mu_{t} \) is an effective turbulent viscosity connecting the wall and the first grid point

\[ \mu_{t} = \frac{y^{+} \mu_{wall} U_{c}}{U_{r} y^{+}} \]  

An advantage of Equation (17) in calculating separated flows is worthy of note: the direction of the wall shear stress \( \tau_{wall} \) is determined by that of the flow velocity \( U \) while \( \tau_{wall} \) calculated from Equation (10) or (1) can only have a positive sign.

From Equation (17), the general form of shear stress at the first grid point can be expressed as

\[ \tau = \tau_{wall} = \lambda U_{t} \]  

where

\[ U_{t} = U_{wall} \]  

[1]{Van Driest, 1951}
\[
\lambda = \begin{cases} 
\frac{\mu_I}{(R_e \Delta n) \Delta \eta} & \text{if } y^+ > 11.6 \\
\mu_{wall}/(R_e \Delta n) & \text{otherwise}
\end{cases}
\]

where \( \lambda \) is the friction coefficient, \( R_e \) is the Reynolds number, \( \Delta n \) is the distance from the wall, and \( y^+ \) is the wall-normal distance. The equations are defined for the near-wall region as a laminar sublayer (\( y^+ < 11.6 \)) and a fully turbulent layer (\( y^+ > 11.6 \)).

\[ \frac{\eta_2^2 + \eta_2^2}{J} = \frac{J-2(\eta_2^2 + \eta_2^2)}{J-1} = \frac{\Delta l^2}{\Delta \Omega} = \frac{\Delta l}{\Delta n} \]
\[ g_{12} = \frac{\xi^2 + \xi^2}{J} \mu_{tot} \frac{\partial U}{\partial \xi} + \ldots \]
\[ g_{13} = \frac{\xi^2 + \xi^2}{J} \mu_{tot} \frac{\partial V}{\partial \xi} + \ldots \]
\[ g_{14} = U \tilde{r}_{11} + V \tilde{r}_{12} + \frac{\xi^2 + \xi^2}{J} \alpha \frac{\partial T}{\partial \xi} + \ldots \]  
and the geometric consideration leads to
\[ \frac{\xi^2 + \xi^2}{J} J^{-2} \frac{\xi^2 + \xi^2}{J} = \frac{\Delta l^2}{\Delta \Omega} = \frac{\Delta l}{\Delta n} \]  
where \( \Delta \Omega \) and \( \Delta l \) are the volume and face area of the control volume, respectively.

After replacing all the stresses acting on the cell face considered by the wall shear stress given by Equation (19), Equations (31) become:

for the west wall
\[ g_{12,w} = \Delta l \lambda U_{1z} \]
\[ g_{13,w} = \Delta l \lambda U_{1y} \]
\[ g_{14,w} = U g_{12,w} + V g_{13,w} + \Delta l a(T - T_{wall}) \]

for the east wall
\[ g_{12,e} = -\Delta l \lambda U_{1z} \]
\[ g_{13,e} = -\Delta l \lambda U_{1y} \]
\[ g_{14,e} = U g_{12,e} + V g_{13,e} - \Delta l a(T - T_{wall}) \]

The sequence in which the above equations are solved together with the Navier-Stokes and turbulence equations in the code is as follows:

a. Initialize all field values.

b. Calculate \( \tau_{wall} \) and \( y^+ \) using Equations (19) and (11).

c. Fix the values of \( K \) and \( \epsilon \) at the first grid points using Equations (15) and (16). Solve the turbulence equations.

d. Calculate \( q_{wall} \) using Equation (23).

e. Calculate \( U_c \) and \( U_c^+ \) using Equations (12) and (10).

f. Update \( \mu_t \) using Equation (18).

g. Update \( \alpha \) using Equation (22).

h. Update \( g_{22}, g_{24} \) using Equation (27) or (29); or update \( g_{12}, g_{14} \) using Equation (39) or (34). Solve the Navier-Stokes equations.

i. Return to step b with updated field values.

The sequence of steps b to i are repeated until the calculation converges.

Note that by definition, the turbulent eddy-viscosity \( \mu_t \) is zero at the wall, such as in the case of the low Reynolds number turbulence models. When using the wall functions, Equation (18) introduces the effective turbulent viscosity which is defined at the wall in the turbulence module. Therefore in post-processing, the wall friction coefficient \( C_f \) can be calculated in the same way as for laminar flows, except that the molecular viscosity \( \mu \) is replaced by the turbulent viscosity \( \mu_t \) at the wall.

### 3. Module Usage

The present turbulence module (version 2.0) is written based on the NPARC version 2.2. The following are those parts of the code which may require user's attention when using it.

#### 3.1 Module

To facilitate identification, all the subroutine names in the module start with CM. In order to use the module, the user only needs to call its three subroutines: CMA0, CMALL and CMVRHS, in the NPARC code.

Subroutine CMA0. This subroutine transfers from NPARC to the module the parameters to define flow, geometric and boundary conditions. In addition, it has the following user-specified parameters:

- \( \text{FDEFER} \) — Blending factor in the convection scheme. Its value may vary from 0 to 1 with the limiting value 0 for the first-order accurate upwind and 1 for the second-order accurate HLPA scheme. The solution tends to be more stable, but also more diffusive when this factor is reduced.

- \( \text{JTDMA}, \text{KTDMA} \) — The value of these integer parameters determines whether to use the TDMA solution algorithm in J- or K-direction (1 does; 0 doesn't). Currently, they are set to

  - \( \text{JTDMA}=1 \)
  - \( \text{KTDMA}=1 \)

- \( \text{BDMAX}(i), \text{BDMIN}(i) \) — Upper and lower bounds for the values of \( K \) (i=1), \( \epsilon \) (i=2) and \( \mu_t \) (i=3).
These bounds are introduced for numerical purposes only, that is, to prevent the corresponding turbulence quantities from becoming negative or abnormally large during the solution process. Currently, they are set to

\[
\begin{align*}
BDMAX(1) &= 1.0 \times 10^6 \\
BDMAX(2) &= 1.0 \times 10^6 \\
BDMAX(3) &= 1.0 \times 10^4 \\
BDMIN(1) &= 1.0 \times 10^{-8} \\
BDMIN(2) &= 1.0 \times 10^{-8} \\
BDMIN(3) &= 1.0 \times 10^{-3}
\end{align*}
\]

which should cover a wide range of the physically meaningful values of \( K, \varepsilon \) and \( \mu_t \). It is to be noted that these values are only valid for the non-dimensional turbulence quantities, as defined in the NPARC code.

RELAX(i) — Under-relaxation factors for \( K \) (i=1) and \( \varepsilon \) (i=2). Currently, they are set to

\[
\begin{align*}
RELAX(1) &= 0.8 \\
RELAX(2) &= 0.8
\end{align*}
\]

Should instability occur, try to reduce these values.

Subroutine CMALL. This is the main subroutine to control the solution sequence in the module. The array variable VIST is the turbulent viscosity \( \mu_t \) which is needed in NPARC for calculating turbulent flows. The array variables TE, ED and YPS are \( K, \varepsilon \) and \( y^+ \), respectively, which can be used for post-processing. Normally, there is no need for user to change this subroutine.

Subroutine CMVRHS. This subroutine which is the counterpart of the subroutine VISRHS in NPARC is for introducing the wall function modifications into the right-hand side viscous flux terms. There is no need for user to change this subroutine.

3.2 NPARC

The authors have made all the modifications necessary for NPARC to use the module. The following shows where these modifications are in NPARC. All the alterations are marked between \( C<< \) and \( C>> \) in the code.

Namelist TURBIN. The integer parameter IMUTR2 is used to select the turbulence models in the module with

\[
IMUTR2 = \begin{align*}
101 & \text{ Chien model} \\
102 & \text{ Shih-Lumley model} \\
103 & \text{ CMOTT model}
\end{align*}
\]

A new integer parameter MWALF is introduced to select the near-wall approach with

\[
\begin{align*}
MWALF = 0 & \text{ low Reynolds number approach} \\
1 & \text{ wall function approach}
\end{align*}
\]

Correspondingly, a new statement is added in the include file NPARC.INC:

\[
\text{COMMON/CMOOTT/MWALF}
\]

and in the subroutine TURBIN:

\[
\text{CALL NLGETI('MWALF',MWALF)}
\]

Subroutines FILT1, FILT2, FILTER. An array FAV01 has been introduced into each of these subroutines to eliminate the artificial viscosity in the near-wall region.

Subroutines INITIA, WREST. The model identifier ITURB for each K-\( \varepsilon \) turbulence model in the module is given an integer value greater than 100. To reflect this expanded choice for turbulence models, the read and write statements for the turbulent quantities in these two subroutines are modified as

\[
\begin{align*}
\text{IF(ITURB.EQ.4 .OR.} & \text{ITURB.EQ.5 .OR.} \text{ITURB.EQ.7 .OR.} \\
& \text{ITURB.GT.100) & \text{READ}(2) \text{ or WRITE}(4) \text{ (values of } K, \varepsilon, \mu_t) \\
& \text{& READ}(2) \text{ or WRITE}(4) \end{align*}
\]

Subroutine MUTURB. The subroutines CMA0 and CMALL of the module are called here.

Subroutine STPF2D. The subroutine CMVRHS of the module is called here.

4. Application

Turbulent boundary layer flow over a flat plate with zero pressure gradient was selected as the first test case for code validation. The solution of the wall function approach (WF) was compared with both the experimental data (Exp) of Wieghardt and Tillmann (1951) and the solution of the low Reynolds number approach (LR). The Shih-Lumley model was used in this test. Note that the high Reynolds number version of this model is of the same form as the standard K-\( \varepsilon \) model of Launder and Spalding (1974). Figure 2 shows the flow geometry and boundary conditions used in the calculation. For the wall function approach, grid points were 111x55, the first grid points above the wall had the \( y^+ \) value of 60 and 14 grid points in the x-direction were located before the leading edge of the flat plate. For the low Reynolds number approach, grid points were 111x81 with the distribution of x-points being the same as in the wall function case and the first \( y^+ \)
ible flows, a freestream Mach number of 0.2 was being 0.3. Since the NPARC code is for compressible flows while the experiment (Wieghardt and Tillmann, 1951) to be compared was for incompressible flows, a freestream Mach number of 0.2 was chosen. The reference Reynolds number $R_e$ was $6.9 \times 10^6$. All calculations started from an initial field with $U = 0.2, V = 0, \mu_s = 1, K = 0.005$ and $\epsilon = 0.09 R_e \rho K^2/\mu_s$. The influence of the artificial viscosity originally implemented in the NPARC code was first examined. This was done by simply reducing the coefficient DIS4 of the 4th-order artificial viscosity. For this particular flow, the 2nd-order artificial viscosity was inactive due to the zero pressure gradient condition. The results are shown in Figure 3 for a streamwise mean velocity profile at a given location and in Figure 4 for the wall friction coefficient. It can be seen that the artificial viscosity with DIS4=0.6 totally spoiled the solution of the wall function approach and very good results were obtained only when DIS4 was reduced to 0.001. Although reducing the artificial viscosity can significantly improve the prediction in this test case, this method is generally unfeasible, because the artificial viscosity will also overly be reduced in places where certain amount is needed. In the near-wall region, the turbulent viscosity prevails, which is enough to stabilize the calculation. Satisfactory results were obtained, as shown in Figures 5 and 6, when turning off the artificial viscosity only in the near-wall area while having DIS4=0.6 elsewhere. Regarding the computational cost, 1000 iterations on the Cray YMP computer took 177 seconds for the wall function approach and 262 seconds for the low Reynolds number approach.

5. Conclusions

The compressible wall functions based on the Van Driest transformation have been implemented in the turbulence module developed for the NPARC code. The details of the numerical implementation are reported.

The module is validated against the incompressible flow over a flat plate. Very good results have been obtained. It is found that the unrealistic artificial viscosity near the wall has a big influence on the solution of the wall function approach. The simplest way to eliminate this influence, as used in this work, is to turn off the artificial viscosity in the near-wall region. The module has to undergo further tests before fully establishing its computational capability.

This work is only restricted to the simple form of the wall functions. Refined forms have been available which account for pressure gradient effects and mass flow through the wall or the effects of the viscous sublayer. Their implementation will be a subject of further development of the module.

6. References

Fig. 1 A typical control volume centered at node C and its neighbouring nodes.

Fig. 2 Geometry of solution field.
Fig. 3 Influence of artificial viscosity on U-velocity profile

Fig. 4 Influence of artificial viscosity on wall friction coefficient

Fig. 5 U-velocity profile

Fig. 6 Wall friction coefficient
# An NPARC Turbulence Module With Wall Functions

## ABSTRACT (Maximum 200 words)

The turbulence module recently developed for the NPARC code has been extended to include wall functions. The Van Driest transformation is used so that the wall functions can be applied to both incompressible and compressible flows. The module is equipped with three two-equation $K\text{-}\varepsilon$ turbulence models: Chien, Shih-Lumley and CMOTT models. Details of the wall functions as well as their numerical implementation are reported. It is shown that the inappropriate artificial viscosity in the near-wall region has a big influence on the solution of the wall function approach. A simple way to eliminate this influence is proposed, which gives satisfactory results during the code validation. The module can be easily linked to the NPARC code for practical applications.

## SUBJECT TERMS

- Turbulence modeling
- Wall functions
- Compressible flow
- Incompressible flow