Report as of FY2007 for 2006FL146B: "Complex flows through culvert structures by CFD modeling"

Publications

Project 2006FL146B has resulted in no reported publications as of FY2007.

Report Follows

FLOW AROUND CULVERT STRUCTURES

Celalettin Emre Ozdemir and Tian-Jian Hsu

Civil and Coastal Engineering
University of Florida

Summary

Culvert is a control structure to convey streamflow through obstructions such as highway embankments. Culvert structures are important because they are control structures and act as boundary condition for large scale events such as flood mapping and mitigation. With the growth of environmental concerns, more accurate prediction on flow through culverts structures becomes necessary in the evaluation of, for example, the contaminants conveyed through it. Therefore the design and the discharge evaluation has become an important issue. Existing rating curves for flow through culverts contain empirical parameters which makes these evaluations with more stringent criterions questionable. With the advances in the computer technology and turbulent flow modeling, it possible to use computational fluid dynamics (CFD) as a tool to evaluate 3-D environmental flows with a variety of turbulence closures. However, CFD's potentials and limitations as a design tool for the flow field around the culvert structures have not been evaluated yet.

In this study, we report a literature survey on the culvert hydraulics for ungated and gated culverts including its theory, field and experimental studies. The applicability of CFD has been evaluated on the basis of what kind of errors it might contain with specific focus on turbulence modeling. Finally, we recommend an integrated approach, using CFD as a primary tool, with an aim to improve upon the existing algorithms to generate rating curves:

Considering the length scale of the problem, we recommend to utilize Reynolds-averaged approach as our primary tool to model the 3D flow field through the culvert structure. The CFD model results can directly resolve detailed flow field and energy dissipation in various components of the culvert structure. The CFD model results can be further analyzed to calibrate various empirical coefficients in the 1D hydraulics equations for culverts, which are difficult to measure directly in the field. The new approach proposed here may improve upon the existing rating curves generated based on lumped method.

From the view point of turbulence modeling, there are concerns regarding the accuracies of the existing Reynolds-averaged approach for complex flow condition. We acknowledge this problem and propose to conduct several detailed turbulent flow simulation using Large-eddy-simulation (LES) to evaluate the accuracy of various turbulence closure scheme in the Reynolds-averaged approach.

Accurate field data remains to be necessary in any model development study. Existing field data obtained by the District is useful for our model-data calibration at the overall culvert system level. However, more detailed flow field data, including turbulence quantities, is highly desirable for a reliable model development.

1.Introduction

A culvert is a conduit that evacuates the streamflow through flow obstruction such as roadway and embankment. Although designed under several considerations such as service time, economy and structural stability, most importantly from the hydraulics point of view it should convey the flow as efficient as possible. Hence within the design limits, flooding at the upstream of the culvert can be avoided. Different types of culverts with various shapes (e.g., elliptic to box culverts, Normann et al. (1985)) and hydraulic features such as tapered inlets are employed in the design process.

Apart from its practical purposes described above, culverts are control structures of a large-scale fluvial system. In other words we can obtain discharge-headwater relationship, i.e. rating curves in the culverts. The rating curves are critical parameterizations of local fluid flow processes in estimating the extreme events in a large scale river system such as flood mapping and flood mitigation. Therefore, the correct estimate of the relation between the headwater and the discharge is critical. For example to determine the flood map of the Lower Deer Creek, numerical study was carried out by using UnTRIM code (MacWilliams et al. 2004). In the aforementioned study there were three culvert boundaries in addition to five others which are either hydraulic structures or gauging stations. The necessary rating curves are developed by means of HEC-RAS using 1-D hydraulic equations. Any errors pertaining to these calculations might propagate throughout the river system. Therefore, the accuracy of HEC-RAS calculations which utilizes the FHWA (Federal Highway Administration) formulae affects the reliability of the large scale computational results.

Existing studies on the flow around the culverts are mostly design oriented. Most of these studies were classical modeling of culverts without the need to resolve the 3-D and unsteady turbulent flow characteristics. Though recently there are some model studies to seek for answers to the fish passage design in the culvert and the performance of the culverts against lock and dam operation. This report is organized as follows. Section 2 reviews the classical approach and design of culvert structures for both gated and ungated and the lessons learned from them. The third section discusses using the computational fluid dynamics (CFD) approach and its reliability with respect to sources of errors and applicability to complex domains. This report is concluded with some recommendations.

2. Theory of Culvert Flow

The flows through the culverts are affected by the flow conditions at the inlet, geometry of culvert, headwater (HW), tailwater and the flow conditions at the outlet (Gonzalez, 2005). Taking all these into account the flow conditions can be classified as inlet controlled flow and outlet controlled flow. This classification is made on the basis of the following fact: if the control section is at the end of the inlet, it is called as inlet controlled flow; while if the control section is at the outlet end or further downstream it is outlet controlled flow. Possible flow patterns for each case are illustrated in Figure 1. According to Figure 1, there is a critical flow formation at the inlet for inlet controlled flow. On the other hand, for the outlet controlled flows there is either a critical flow formation at the outlet or the flow is submerged at both ends.

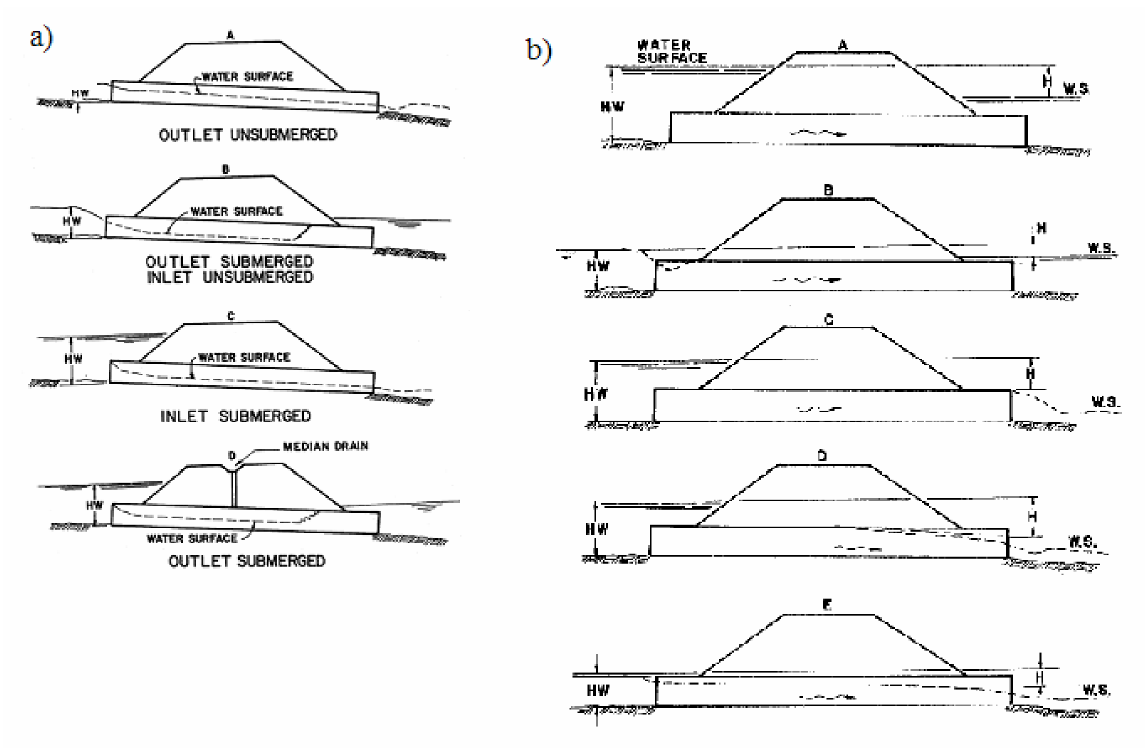

Figure 1 Inlet (a) and the outlet (b) controls of uncontrolled free culverts (adopted from Normann et. al., 1985).

For the inlet control with upstream unsubmerged, flow is treated as the flow through a weir as the flow becomes critical at the inlet. The flow becomes like a weir flow in other words the discharge is proportional to the $HW^{3/2}$. On the other hand, for inlet controlled flow with upstream submerged, flow becomes like an orifice i.e. discharge is proportional to $HW^{1/2}$. For the outlet control the flow is calculated on the basis of the energy balance equations and again the discharge is proportional to $HW^{1/2}$.

Equations (1.1) to (1.3) are used in inlet control culverts with equation (1.3) specifically used for the submerged inlet condition.

$$\frac{HW_i}{D} = \frac{H_c}{D} + c[\frac{Q}{AD^{0.5}}]^2 + Y - 0.5S \quad (1.1)$$

$$\frac{HW_i}{D} = K[\frac{Q}{AD^{0.5}}]^M \quad (1.2)$$

$$\frac{HW_i}{D} = c[\frac{Q}{AD^{0.5}}]^2 + Y - 0.5S \quad (1.3)$$

where

D : interior height of the culvert barrel,ft
H_c : specific head at critical depth
Q : discharge through the culvert
A : full cross-sectional area of the culvert
S : culvert barrel slope
K, M, c, Y: constants dependent on the shape and entrance.

$$Z_3 + Y_3 + \frac{\alpha_3 V_3^2}{2g} = Z_2 + Y_2 + \frac{\alpha_2 V_2^2}{2g} + H_L \quad (1.4)$$

where

Z_3 : Upstream invert elevation of the culvert
Y_3 : The depth of water above the upstream culvert inlet
V_3 : The average velocity upstream of the culvert
α_3 : The velocity weighting coefficient at the upstream of culvert
g : gravitational acceleration
Z_2 : Upstream invert elevation of the culvert
Y_2 : The depth of water above the upstream culvert inlet
V_2 : The average velocity upstream of the culvert
α_2 : The velocity weighting coefficient at the upstream of culvert
H_L : Total energy loss through the culvert

Examples using these relations between headwater and discharge are calculated and performance curves are plotted in Figure 2. These curves are widely used in the design of the culverts. For both upstream control and downstream control culvert flow computations employ empirical coefficients i.e. the use of *K, M, c, Y* in the upstream control culvert flow and use of Manning's empirical coefficients in the computation of H_L. For the gated culverts as we shall see there are empirical coefficients that vary with the shape and entrance of the culverts in order to accurately calculate the head loss. This raises the question about the reliability and robustness of the existing performance curve equations, especially when more stringent design criterions are expected to be met due to environmental concerns.

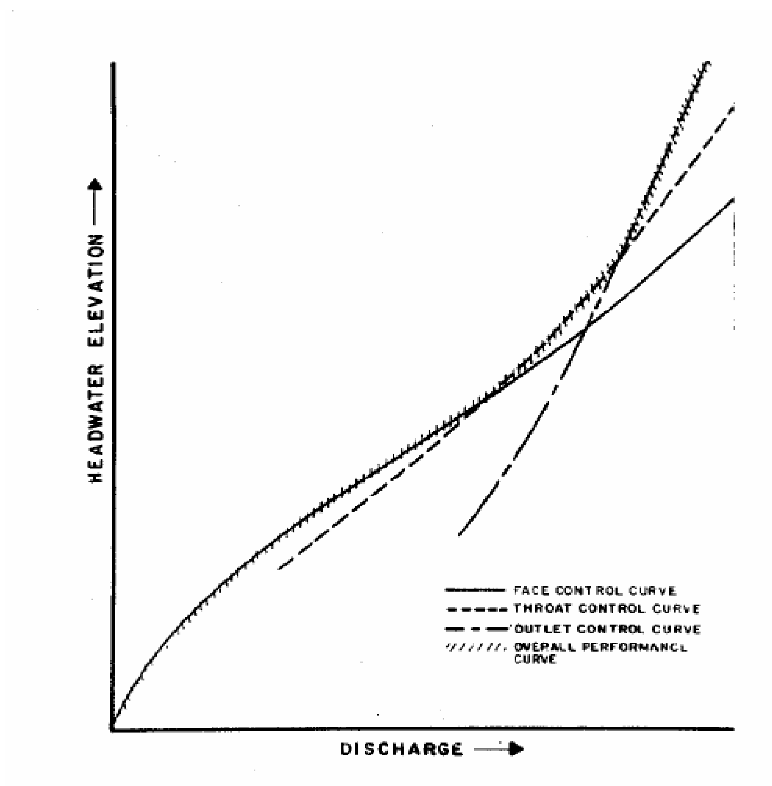

Figure 2 Culvert performance curve without overtopping (adopted from Normann et al. 1985).

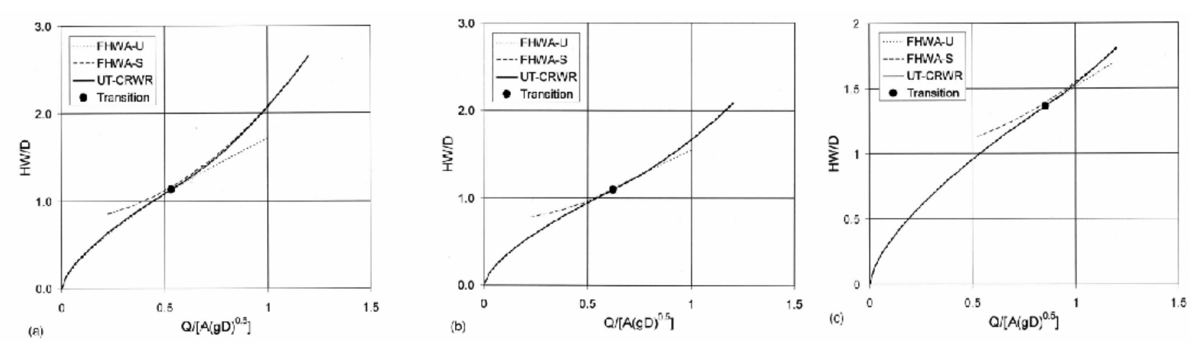

Figure 3 Reevaluation of the performance curve by Charbeneau et al. (2006).

There are studies attempt to reduce the number of empirical coefficients. In Charbeneau et al. (2006), the number of empirical coefficients is reduced from four to two, including the contraction coefficient and the soffit contraction coefficient. The HW relation from culvert performance and discharge is equalized and solved for HW. Also the singular point at the transition between the submerged and unsubmerged inlet control flow is eliminated as the derivatives of both relations are found to be equal from the aforementioned relation, giving a more flexible tool for design purposes.

Rating algorithms for *gated* culverts are scarce in the literature. One can determine the flow by simplifying the flow a 1-D basic fluid flow. Gonzalez (2005) presented the pressurized fluid flow through two types of gated culverts (Figure 4). In the first type (Figure 4A), the flow is governed by the gate; while in the second type (Figure 4B) the flow is controlled by the weir erected at the upstream of the gate. In the first type, the discharge coefficient is represented in terms of loss coefficients and ratio of the consecutive cross sectional area ratios as well as the frictional losses throughout the culvert pipe which is obtained as a result of Manning's formula proposed by Yen (1992). In the second type the discharge coefficient, due to hardships in monitoring the flow throughout the flow the estimation is obtained based on the weir flow equation. In other words, the discharge coefficient is expressed in terms of the difference between HW and weir elevation at the upstream, weir crest length in the transverse direction.

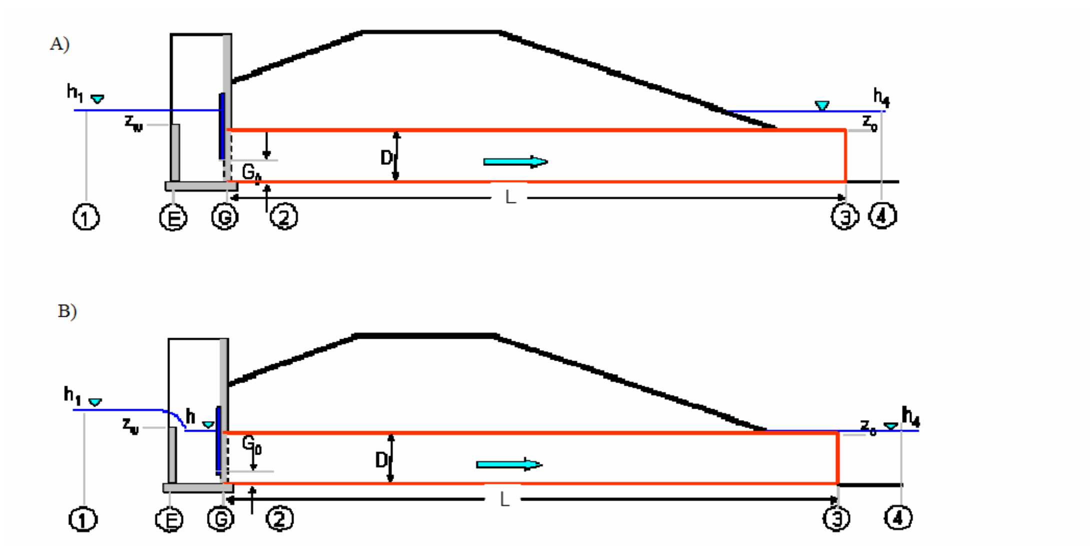

Figure 4 Two flow configurations for pressurized gated culvert flow (adopted from Gonzalez, 2005)

The studies so far, which are primarily based on simplified 1-D equations and field studies, are primarily led by the hydraulics design considerations. The main objective is to find simple relations to make the design time minimal and keeping the results reasonable. However as previously mentioned, any error in the culvert flow may propagate in larger scale and for environmental concern, one requires more stringent level of accuracies in the culvert designs. The contribution potential from 3-D flow analysis has not been assessed for culvert flow in order to minimize and evaluate the error introduced by these simplified equations. In addition, the likelihood of scour, which is a common problem around culverts, can be determined by 3-D turbulent characteristics of the flow and the resulting bed shear stress. Very little has been done to determine the detailed turbulent flow characteristics of culvert flow. Day (1997) investigated flow turbulence at the inlet of a pipe culvert by laboratory experiments. The flow field was measured by means of electromagnetic current meter. Averaged flow field and the turbulent intensity were evaluated near the inlet.

Gonzalez (2005) reports results based on field experiments of flow through culverts structures. The flow is measured by monostatic acoustic Doppler flow meters (ADFMs). Its contrivance is based on two pairs of transducers one to estimate the flow depth and the other to estimate the flow velocities. As its name implies the flow velocity is estimated by the shift in the signals created by the transducer. Afterwards, this study combines the flow data with the depth data and utilizes the no slip boundary condition at the wall.

3.CFD Applications

More accurate and physical-based approaches for developing rating curves require detailed analysis that directly resolves the 3D flow field in the culvert structure. Comprehensive 3D flow analysis can be accomplished by detailed flow measurements and/or utilizing Computational Fluid Dynamics (CFD) as a tool. While detailed flow measurements are essential but often time-consuming and expensive, flow analysis based on CFD has been becoming an effective and economical design tool in various industrial and engineering applications. In this study, we believe it is possible to use CFD analysis to resolve 3D fluid flow and the dissipation around culverts and hence improves upon the rating algorithms based on simple 1D flow equations. The turbulence created at the upstream of the culvert combined with the irregularities of in the flow field has a unique effect in the performance of a culvert. Although in the classical evaluation, irregularities are included in the discharge equations through empirical coefficients, their robustness and accuracy are of question. Therefore it is the authors' belief that it is highly potent that CFD analysis can improve the accuracy and reliability of the discharge coefficient in the rating curves.

Nearly all environmental flows are turbulent flows. Therefore, errors resulted from CFD modeling/simulation for environmental flow are due to the combination of 1) modeling errors, 2) discretization errors, 3) iteration errors and 4) programming and user errors. These aspects are discussed in the following sections.

3.1.Modeling Errors

Turbulent flow is difficult to model because of its random, stochastic and unsteady nature. The most accurate solution of turbulent flow is the direct numerical simulation (DNS) as it solves all the scales of turbulent motion. The use of DNS will not yield any modeling error and hence it is often utilized to calibrate and validate other turbulence modeling approaches (e.g., RANS, or LES approaches). The practical use of DNS is limited due to its requirement of high spatial and temporal resolution and hence CPU time. The spatial resolution should be smaller than the Kolmogorov length scale to resolve the smallest eddy scale. In a three-dimensional space this implies computational requirement that scales with $Re^{9/4}$ (e.g., Pope 2000). In addition, one often needs to compute DNS for significant length of time in order to obtain meaningful turbulence statistics. Hence, currently the use of DNS is limited to lower Reynolds number flow (Bhagangar et al. 2002).

For hydraulics application, the most popular approach is based on solving Reynolds-Averaged Naviers-Stokes (RANS) equations with turbulence closures. The RANS approach is efficient. However, the fundamental assumption of RANS approach implies parameterizations on all the scales of turbulence and hence the inherited turbulence closure problems require careful consideration. Recently, the improved computer power allows using Large-Eddy Simulation (LES) approach for some hydraulic and fluvial applications (e.g., Keylock et al. 2005). Comparing with the RANS approach, LES is more accurate in turbulence closure because most of the anisotropic energy containing turbulent eddies are directly resolved and the sub-grid closure scheme is only required for small scales, which is easier to parameterize. Despite the computational requirements in LES remains significant for high Reynolds number hydraulics flow, it is possible to utilize LES to conduct few detailed simulations to provide flow databases in order to calibrate the turbulence closure scheme in the RANS approach (e.g., Rodi et al. 1997). In this report, we primarily focus on RANS approach and its turbulence closures for engineering applications. However, we will also pursue evaluating the RANS approach using LES and small-scale laboratory experimental data.

The eddy viscosity closure is the most popular scheme for solving the Reynolds Averaged Navier-Stokes equations. In the RANS approach, the instantaneous, intermittent turbulence is averaged out. Although this causes loss of information in the instantaneous flow field, it gives the overall statistical quantities of turbulence which is in fact is the more useful information for most practical engineering applications. Utilizing the eddy-viscosity hypothesis, the most popular model for second order closure scheme is the *k-ε* model. Turbulent kinetic energy *k* and its dissipation rate, *ε*, are the model parameters to calculate the eddy viscosity and the preceding parameters are closed by means of the prognostic equations given as follow:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}[(\mu+\frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_j}]+G_k+G_b-\rho\varepsilon-Y_M+S_k \tag{2.1}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon)+\frac{\partial}{\partial x_i}(\rho \varepsilon u_i)=\frac{\partial}{\partial x_j}[(\mu+\frac{\mu_t}{\sigma_\varepsilon})\frac{\partial \varepsilon}{\partial x_j}]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon \tag{2.2}$$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients. G_b is the generation or damping of turbulence kinetic energy due to buoyancy, Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate, $C_{1\varepsilon}$, $C_{2\varepsilon}$, and $C_{3\varepsilon}$ are constants, σ_k and σ_ε are the turbulent Prandtl numbers for *k* and *ε,* respectively. S_k and S_ε are the source terms.

The drawbacks of this model can be stated as follows: 1) The eddy viscosity hypothesis itself, which assumes complete analogous to Newtonian viscous fluid, is questionable. Empirical damping function is often used in calculating eddy viscosity. 2) The transport equation of *ε* is proposed according to several additional assumptions, including high Reynolds number flow. 3) There exist additional source terms in the prognostic equations of both ε and k, 4) Appropriate boundary condition for *ε* is not always obvious.

Damping functions used in *k-ε* model is used to mimic the effect of the molecular viscosity such as in the case of near wall low Reynolds number flow. In most applications, the near wall regime is not resolved, instead the model is forced by parameterized wall-functions. Although it might seem disadvantageous for the reliability of the solution, the review by Patel et al (1985) demonstrate that careful use of the so-call low Reynolds number modifications to *k-ε* model renders results that agrees reasonably well with the measured data (or LES results). Recently, Goncalves and Houdeville (2001) present the robustness of the wall functions over computational grids ranging from coarse to refined ones.

For different types of flows, the model coefficients in the *k-ε* model are not necessarily unique. One reason is the existence of uncertainties in the experimental data to obtain these coefficients, which are usually conducted in highly simplified and idealized conditions. Another reason is, complementary to the first reasoning, the dependence of these coefficients to each other in complex flow. The first reasoning is explained in more detail as the following. Each parameter in the equations controls specific transport mechanism and can be obtained by simplified flow conditions that isolate the desired mechanism. For example, $C_{\varepsilon 2}$ can be obtained from the decaying homogeneous turbulent flow. In practice, this is approximated as grid generated turbulence as the other terms vanish in the prognostic equation of the dissipation. However the result of the experimental data is not unique but falls onto a range of values. Although after several experiments and trials for different type of flows, $C_{\varepsilon 2}$ is found to be 1.83, on the physical basis it is not possible to assign a unique value. Another coefficient, $C_{\varepsilon 1}$ is the coefficient responsible for the dispersion of the free shear layers. The growth rate for homogeneous flows is found to be a function of $C_{\varepsilon 2}$- $C_{\varepsilon 1}$ which shows the dependency of both parameters. If the values of $C_{\varepsilon 2}$ and $C_{\varepsilon 1}$ have uncertainties, subsequent calibrations on other parameters are also inevitably uncertain. The experiment of $C_{\varepsilon 1}$ is found to fall onto a range with an upper limit 1.51. However 1.44 gives reasonable results (Durbin and Reif, 2000). Also in the case of high Re, C_μ is not calculated correctly although it does not change the mean flow field as the log-layer eddy viscosity is calculated correctly (Durbin and Reif, 2000).

The *k-ω* model is similar to the *k-ε* model and *ω* is the ratio of the dissipation rate to the TKE of turbulence, which has a dimension of inverse of time. Although very similar at glance, the difference between these two parameters becomes obvious in the log layer of the wall for wall bounded flows. For non-homogeneous flows a diffusion term should exist in addition to the homogeneous condition. However, the addition of diffusion term to the homogeneous solution fails as it results in a negative diffusion of time in the log layer. The use of a (inversed) timescale for the second parameter in the two-equation closure might be an alternative to resolve this problem. For this, the use of ω becomes a plausible alternative because ω represents the inverse of a timescale. In fact, one can re-exam this issue by deriving the *ε*-equation from the ω-equation. The resulting *ε*-equation has an extra cross-diffusion term as compared to the original *ε*-equation (e.g., Wilcox 1993).

Despite the time-scale concept used of k-ω may be more meaningful, the k-ω model is not free of errors. Its major drawbacks are 1) it over predicts the shear stress in adverse pressure gradient boundary layer (Menter, 1994) 2) it produces unreliable results in free shear layer flows hence not sensitive to free-stream conditions. Menter (1994) argued that basic shortcomings of the k-ω can be partly avoided by using two layer approach i.e. use k-ω model in near wall region and k-ε model elsewhere which is also known as SST k-ω model.

RNG based k-ε model is first proposed by Yakhot and Orszag (1986).This model is based on the assumption that the length scales for small eddies are approximated by the Kolmogorov energy spectrum. In the aforementioned model turbulence is created by the assumed random force between two points and any fluctuating variable can be obtained in this manner. However $C_{\varepsilon 1}$ is obtained as 1.063 which yields high TKE growth rate in shear flows compared to physical and numerical experiments (Speziale et. al., 1989). Therefore the model is then modified by cutting the random force for small wave numbers. The equations are then reformulated and the coefficients are reevaluated. As a result the $C_{\varepsilon 1}$ and $C_{\varepsilon 2}$ changed while the remaining stayed the same (Yokhat et. al., 1992).

Nonlinear Reynold's stress model (RSM) is different from eddy viscosity models as additional non-linear terms are added to the eddy viscosity model. In scalar eddy viscosity approach, direction of the mean strain rate and the Reynolds stress are forced to be aligned which is true for pure strain but the flow with mean vorticity. In RSM models in the literature, anisotropy of the turbulent flow has been tried to be modeled. Basically this has been done in two basic approaches. In the first one nonlinear Reynolds stress model is to be developed by employing tensorial viscosity and in the second one the prognostic equations are developed for each Reynolds stress. The former approach has been modeled in three major ways: 1) modeling anisotropic turbulence similar to laminar non-Newtonian flow (Spaziale, 1987) 2) use of statistical approaches (Nisizima and Yoshizawa, 1987) 3) use of RNG theory (Rubinstein and Barton, 1990). In the first one the modeling of non-Newtonian flow is analogous to modeling of Reynold's stress as stress-strain relation is nonlinear and the stresses depend on the mean flow quadratically. In the second case, turbulence is regarded as a phenomenon with universal and non-universal behavior which is the result of either geometry or boundary condition. Therefore the affect of universal behavior on the non-universal one is calculated statistically. This approach yield satisfactory results in square duct flow. In the third case the flow use of RNG yields the Reynolds stresses as power series of a parameter. While the first order simply yields the eddy viscosity model, the second order yields a quadratic nonlinear model. The disadvantage of the first two approaches is that they only contain algebraic equations alone transport effects and in both convection and diffusion. Therefore they can be only applicable to flows where the dissipation and production of TKE is equal.

Though the RSM models heretofore tried to account for the anisotropy of turbulence it fails to predict the return the isotropy after the removal of the strain. To estimate the correct the mentioned deficiency can be overcome by exactly state the nonlinear equation and reinterpret the terms physically and model the terms. The following is the rearranged version of the exact Reynolds stress transport equation;

$$\frac{D\overline{u_i u_j}}{Dt} = (D_{ij}^t + \Phi_{ij,p} + D_{ij}^{\upsilon}) + P - \varepsilon_{ij} + (\Phi_{ij,1} + \Phi_{ij,2}) \tag{2.3}$$

Where D_{ij}^t and D_{ij}^v are the turbulent and viscous diffusion, respectively, $\Phi_{ij,p}$ is the pressure diffusion, P is the production, ε_{ij} is the dissipation, $\Phi_{ij,1}$ and $\Phi_{ij,2}$ are the is the slow and rapid pressure strain.

In equation (2.3) production term and viscous diffusion need no modeling. Pressure diffusion added to turbulent diffusion, dissipation and pressure strain terms are modeled. These models are given in Tables (1) through (4). Hanjalic and Launder (1972) proposed a model that excludes the pressure diffusion term due to experimental results which shows that pressure diffusion is small compared to turbulent diffusion. Daly and Horlow (1970) proposed a simple gradient diffusion model including pressure diffusion. Chen even proposed a simpler model treating each Reynolds stress term as a fraction of TKE.

Table 1 Modeled Reynolds Stress Equation for diffusion term

	Equation
Daly and Harlow (1970)	$\frac{\partial}{\partial X_k}[C_k \frac{k}{\varepsilon} \overline{u_i u_j} \frac{\partial \overline{u_i u_j}}{\partial X_l}],\ C_k = 0.25$
Hanjalic and Launder (1972)	$\frac{\partial}{\partial X_k}[C_k \frac{k}{\varepsilon} (\overline{u_i u_l} \frac{\partial \overline{u_j u_k}}{\partial X_l} + \overline{u_j u_l} \frac{\partial \overline{u_j u_k}}{\partial X_l} + \overline{u_k u_l} \frac{\partial \overline{u_j u_k}}{\partial X_l})],\ C_k = 0.11$
Chen (1983)	$\frac{\partial}{\partial X_k}[C_k \frac{k^2}{\varepsilon} \frac{\partial \overline{u_i u_j}}{\partial X_l}],\ C_k = 0.09$

Table 2 Modeled Reynolds Stress Equation for dissipation term

Rotta (1951)	$\frac{\overline{u_i u_j}}{k} \varepsilon$
Hanjalic and Launder (1972)	$\frac{2}{3}\delta_{ij}\varepsilon$
Launder (1986)	$\varepsilon_{ij} = \frac{\varepsilon}{k}(\overline{u_i u_j} + \overline{u_i u_j}\, n_k n_j + \overline{u_j u_k}\, n_k n_l + \delta_{ij}\, \overline{u_k u_l}\, n_k n_l)/(1 + \frac{5}{2}\frac{\overline{u_p u_q}\, n_p n_q}{k})$
Fu et. al. (1987)	$f_\varepsilon \frac{2}{3}\delta_{ij}\varepsilon + (1 - f_\varepsilon)\varepsilon_{ij}^w;\ f_\varepsilon = A^{1/2};\ A = [1 - \frac{9}{8}(a_{ij}a_{ij} - a_{ij}a_{jk}a_{ki})];\ a_{ij} = (\frac{u_i u_j}{k} - \frac{2}{3}\delta_{ij})$

Table 3 Modeled Reynolds Stress Equation for slow pressure strain term

Rotta (1951)	$\Phi_{ij,1} = -C_1 \frac{\varepsilon}{k}(\overline{u_i u_j} - \frac{2}{3}\delta_{ij} k)$
Lumley and Khajeh-Nouri (1973)	$\Phi_{ij,1} = -(C_1 + C_1{}^n a_{ij} a_{ij})\varepsilon a_{ij} - C_1{}^l \varepsilon (a_{im} a_{jm} - \frac{1}{3}\delta_{ij} a_{ij} a_{ij})$

For the dissipation modeling, at high Re the small scale eddy structures are isotropic and the model proposed by Hanjalic and Launder (1972) can be used without affecting the solution. For the flows of low Re such as near wall Rotta (1951) underestimates the dissipation. Launder (1986) proposed a complex model which adopts the dissipation the same as Hanjalic and Launder (1972) and correct it with pressure-strain correlation. The model proposed by Fu et al. (1987) makes the correction by a function of which parameter is the difference between the isotropic turbulence and anisotropic one.

The model for the slow pressure-strain term there is no agreement in values of the constants. However the highest of two considers the slow pressure-strain term as the contributing term which might be possible for a narrow range of turbulent flows. However there is no acceptable model.

Table 4 Modeled Reynolds Stress Equation for Rapid Pressure Strain Term

Hanjalic and Launder (1972)	$\Phi_{ij,2} = a_{ij}{}^{ml}(\frac{\partial U_m}{\partial X_l} + \frac{\partial U_l}{\partial X_l})$; $a_{ij}{}^{ml} = \alpha\, \overline{u_m u_i}\, \delta_{ij} + \beta(\overline{u_m u_i}\, \delta_{ij} + \overline{u_m u_j}\, \delta_{il} + \overline{u_i u_i}\, \delta_{mj}) + (\gamma \delta_{mi}\delta_{ij} + \sigma[\delta_{mi}\delta_{ij} + \delta_{mj}\delta_{il}])k$; $+C_2\, (\overline{u_m u_i}\, \overline{u_l u_j}) / k + \nu(\overline{u_m u_j}\, \overline{u_i u_l} + \overline{u_m u_l}\, \overline{u_i u_j}) / k$ $\alpha = (10 - 8C_2)/11$; $\beta = -(2 - 6C_2)/11$; $\nu = -C_2$; $\gamma = -(4 - 12C_2)/55$; $\sigma = (6 - 18C_2)/55$; $C_2 = 0.45$
Launder et. al. (1975)	$\Phi_{ij,2} = -\frac{C_2 + 8}{11}(P_{ij} - \frac{2}{3}\delta_{ij} P) - \frac{30C_2 - 2}{55} k(\frac{\partial U_m}{\partial X_l} + \frac{\partial U_l}{\partial X_l}) - \frac{8C_2 - 2}{11}(D_{ij} - \frac{2}{3}\delta_{ij} P)$ $P_{ij} = (\overline{u_i u_k}\frac{\partial U_j}{\partial X_k} + \overline{u_j u_k}\frac{\partial U_i}{\partial X_k})$; $P = -\overline{u_i u_j}\frac{\partial U_i}{\partial X_j}$; $D_{ij} = -(\overline{u_i u_k}\frac{\partial U_k}{\partial X_j} + \overline{u_j u_k}\frac{\partial U_i}{\partial X_i})$; $C_2 = 0.4$;
Gibson and Launder (1978)	$\Phi_{ij,2} = -\xi(P_{ij} - \frac{2}{3}\delta_{ij} P)$; $P_{ij} = (\overline{u_i u_k}\frac{\partial U_j}{\partial X_k} + \overline{u_j u_k}\frac{\partial U_i}{\partial X_k})$; $P = -\overline{u_i u_j}\frac{\partial U_i}{\partial X_j}$; $\xi = 0.6$

Fu et. al. (1987)	$\Phi_{ij,2} = 0.6(P_{ij} - \frac{2}{3}\delta_{ij}P) + 0.3\varepsilon a_{ij}(\frac{P}{2\varepsilon}) - 0.2\{\frac{\overline{u_k u_j u_l u_i}}{k}[\frac{\partial U_k}{\partial X_l} + \frac{\partial U_l}{\partial X_k}] - \frac{\overline{u_l u_k}}{k}[\overline{u_i u_k}\frac{\partial U_j}{\partial X_l} + \overline{u_j u_k}\frac{\partial U_i}{\partial X_l}]\} - 0.6[A_2(P_{ij} - D_{ij}) + 3a_{ml}a_{nj}(P_{mn} - D_{mn})]$; $A_2 = a_{ij}a_{ji}\,;\, a_{ij} = (\frac{\overline{u_i u_j}}{k} - \frac{2}{3}\delta_{ij})$

For the rapid pressure-strain term Hanjalic and Launder (1972) it does not satisfy a kinematic boundary condition and give a wide range of results if there exists a complex strain field. The model proposed by Gibson and Launder (1978) does not predict the effects of swirl on the spreading rate of and axisymetric jet (Launder and Morse, 1979). Generally speaking, return to isotropy is slower with these models, although they might work under different flows (Jaw and Chen, 1997).

Finally, we would like to comment on the Large-eddy simulation (LES) approach. From the computational point of view, LES is somewhat the compromise between DNS and RANS models. It is computationally less expensive than DNS but more costly than RANS, on the other hand history of the flow is preserved. As the large scale motions are more energetic they carry most of the energy and transport of the conserved properties. Likewise LES resolves the energetic structures in the flow and model small ones. The large scale components are obtained by filtering like box filtering, Gaussian filtering, cut-off filter in which above certain value of wave numbers are discarded. The filter length is denoted as Δ. After rewriting the Navier-Stokes equations and subtracting from the RANS equations the result yields us the sub-grid scale Reynold's stress;

$$\tau_{ij}^{s} = -\rho(\overline{u_i u_j} - \overline{u}_i\,\overline{u}_j) \tag{2.4}$$

This term is modeled by different sub-grid scale models. Smagorinsky models, dynamic models and deconvolution models. Smagorinsky model can be stated as follows;

$$\tau_{ij}{}^{s} - \frac{1}{3}\tau_{kk}{}^{s}\delta_{ij} = \mu_t(\frac{\partial \overline{u}_i}{\partial \overline{x}_j} + \frac{\partial \overline{u}_j}{\partial \overline{x}_i}) = 2\mu_t\,\overline{S_{ij}} \tag{2.5}$$

$$\mu_t = C_S{}^2\rho\Delta^2\,|\,\overline{S}\,| \tag{2.6}$$

Model parameter C_s is usually found from the experiments to be around 0.2. On the other hand for different flow types this value is different as it is a function Re and other non-dimensional flow parameters (such as in stratified flows Richardson or Froude numbers). In shear flows and near the wall the change of this parameter is necessary. To overcome the near wall problem possible recipes are 1) using VanDriest damping function, 2) reduce the eddy viscosity when sub-grid Re is small (Mac-Millan and Ferziger, 1980) 3) use of RNG theory (Yakhot and Orszag, 1986).

In the dynamic models it is assumed that the largest subgrid scale motions can be modeled by smallest resolved scale motion. Broader filter is used to get a very large scale field and an effective subgrid scale is obtained. Reynold's stress is therefore estimated by this procedure for every point at every time. Consequently the model parameter is produced in a consistent manner and the problems like near wall and anisotropy removed. However model parameter is a rapidly varying function of spatial coordinates and time which leads to high values of model parameter in both signs. If this recurs for a considerable time over a considerable range this causes numerical instability.

Deconvolution method tries to find the sub-grid scale velocities by the filtered ones. Unfiltered velocities are represented by Taylor series expansion. If the series cut off at most up to second order the differential equation for the unfiltered velocity is obtained in terms of filtered ones.

3.2.Discretization Errors

Most of the time governing equations in the fluid flow cannot be solved exactly. The algebraic set of equations to be solved, are obtained as a result of approximations. Evaluation of volume and surface integrals in finite volume and numerical calculation of derivatives are basically obtained after approximations. The details of the numericals will not be given here, however an overview will be given.

Upwind difference scheme (UDS), central difference scheme (CDS), quadratic upwind interpolation for convective kinematics (QUICK) are the existing schemes in most of the commercial CFD codes (for details refer to any textbook regarding this subject). Therefore careful employment of these schemes is crucial for accuracy, simplicity and efficiency. According to the findings of Ferziger and Peric (2001) higher order schemes converge to an accurate result although oscillating on the coarse grids. First order UDS should be avoided as it introduces error function as a diffusion equation which smears out the error especially in 3-D simulations. CDS gives the best compromise regarding efficiency, simplicity and accuracy. Apart from these it is also worth to note that the discretization error introduced on non-uniform grid is proportional to stretching ratio (r) minus unity and it is amplified where there is a strong variation in the flow.

3.3.Iteration Errors

After the discretization there occurs a set of non-linear equations which usually further linearized. The direct solution of these equations is costly and usually they are solved by numerical methods. There is no way that the exact solution can be obtained as a result of these equations hence the solutions are stopped based on the convergence criterions. The criterions are generally set to one order less than the discretization error.

Therefore these types of errors have the least effect on the solution. The details of these can be found in any book related to CFD.

3.4.Programming and User Errors

The most important of all is the boundary condition errors and errors due to poor quality input grid. The flow should essentially reflect the actual conditions of real flow. Most of the time, it is not possible to implement the exact flow boundary conditions and geometrical configuration of the flow field. This may stem either from the solver program or the approximations which has to be made in the flow field. Any error made in boundary conditions propagates –the speed of which depends on the computational scheme employed- in the flow field and undermines the reliability of the solution. This holds true also for the input grid. One should intuitively determine the flow field and reckon the fields where there is high resolution required for the grid generation. Apart from the resolution determination, the topology of computational cells is important in the sense that the accuracy of the central difference scheme is more accurate in quadrilateral and hexahedral cells than the ones in the triangular and tetrahedral elements. Any grid with any topology should satisfy quality constraints which are based on geometric properties of the computational cell such as skewness, stretching ratio (r) of adjacent cells and aspect ratio. The circumcenter of highly skewed elements lies outside the boundaries of the computational cell. Since the pressure and any scalar are stored at the circumcenter, inaccurate results would be obtained for pressure gradients in highly skewed elements. If the aspect ratio of the computational cell is higher than there occur problems in the approximation in the diffusive fluxes which is a major concern in the near wall (Ferziger and Peric (2001)). The use of non-uniform grid is nearly unavoidable in complex flows. Therefore the discretization error, as it has been mentioned in section 3.2 is proportional to r-1. Hence the use of grids with r greater than 2 especially in the regions with high velocity gradients amplifies the discretization error. It should be also mentioned that the grid generation depends on the turbulence model used i.e LES requires higher resolution than RANS models but less than direct numerical simulation (DNS). In LES, one often needs to check the model results for resolved energy field or the calculated energy spectrum to ensure that appropriate grid resolution is adopted.

4. Recommendations

Although currently there is no detailed field data available, it is the authors' belief that CFD is a potential tool to understand the physics of the flow field. As stated in the previous section, the RANS approach is less expensive compared to LES and DNS. Despite several existing criticisms on the limitation of eddy-viscosity-based two-equation approach for complex flows, the use of more appropriate closure model (less restrictive assumptions) such as RSM and RNG models shall yield more accurate results. Also in RANS approach, the grid generation will require less time as compared to the LES grid generation.

Because the limited field data and the scale involved in the prototype culvert structure, the feasibility of using LES for CFD computation cannot be assessed with confidence at this point. In addition, the high quality input grid (geometry of the culvert structure) should be assured. This restriction may sometimes cause the application of LES nearly impossible to implement, such as the geometry of draft tube in the hydropower structures. Even if detailed geometry is available, it often takes a great amount of time to make the sensitivity analysis of the input grid. Therefore the applicability of LES should be investigated with the actual geometric data and the flow conditions. On the other hand, the payback of the LES employment is large because it theoretically gives more accurate results. Especially in the lack of field and laboratory data, LES results can be used as the computational experiment tool in complex field configuration (in which DNS cannot be used, See Constantinescu et. al., 2006). Consequently if accurate LES results can be obtained, one can conduct useful inter-comparison among the turbulence models and the optimum one among them could be evaluated in terms of computational expense and accuracy and eventually used as design tool. With the exiting computational power in the water resource group at University of Florida, we shall be able to employ few LES parallel computations using the new 20-processor cluster system. These few LES results can provide as valuable database to evaluate various RANS approaches.

As it is previously mentioned, the accuracy of CFD simulations depends on the quality of the input grid and the boundary conditions. The flow characteristic of a culvert flow is unique as the upstream boundary conditions differ with the topography of the flow field. This is because any perturbation to the flow field upstream will be conveyed to downstream which also affects the flow through the culvert. However the degree of uniqueness of the flow can be evaluated for the same structure at different locations. This necessitates apart from the structural details of the culvert data, the actual topographic and flow data at different culvert locations (can you re-phrase, I am not getting it). In addition if the real time flow field data is available, the turbulence closure models should be directly tested.

The study of Gonzalez (2005) shows that for the gated-culvert flow, there are various empirical loss coefficients exist to parameterize energy dissipation at different component of a culvert system. Due to the high complexity of these energy dissipation mechanisms, Gonzales (2005) employed lumped approach in which the details of these coefficients were not evaluated while an bulk discharge coefficient is defined in terms of dimensionless numbers. These bulk coefficients are then evaluated by fitting a best-fit curve with respect to the ratio of the cross sectional areas of gate opening to the cross sectional area of the culvert barrel. The lumped approach employed by Gonzalez (2005) is plausible because there is no available loss coefficient data for each component in the culverts in the literature and it is not easy to directly measure the flow in these kinds of structures. We believe that with the use of CFD, it it possible to evaluate each of these loss coefficients carefully. In addition to these empirical parameters, the momentum correction factor, α_1, used in the inlet weir controlled structure, should also be evaluated numerically with CFD, which will also include the 3-D affect. All the CFD results along with available field data may then be used to re-formulate the rating curves which shall improve upon the existing one based on lumped approach.

References

Bhagangar, K., Rempfer,D., and Lumley,J.L., 2002, Direct numerical simulation of spatial transition to turbulence using fourth-order vertical velocity second-order vertical vorticity formulation, *J. Comp.Phys*.180:200-228.

Charbeneau R. J, A. D. Henderson, and L. C. Sherman (2006), Hydraulic Performance Curves for Highway Culverts, J. Hyd. Engrg., ASCE 132(5), 474-481.

Constantinescu, S.G., W.F. Krajewski, C. E. Ozdemir, T. E. Tokyay (2006), Simulation of Airflow around rain gauges: Comparison of LES with RANS models, Advances in Water Resources (in press).

Daly, B. J., and Horlow, I. H. (1970), Transport Equations in turbulence, Phys. Of Fluids, 13, 2634-2649.

Day R. A.(1997), Preliminary Observations of Turbulent Flow at Culvert Inlets, J. Hyd. Engrg., ASCE 123(2), 116-124.

Durbin, P.A.,B.A.P. Reif, Statistical Theory and modeling for Turbulent Flows, Wiley, Stanford, C.A.,2000.

Ferziger, J. H., M. Perić, Computational Methods for Fluid Dynamics, 3rd Edition, Springer, Berlin, 2002.

Fu, S, Launder, B. E., and Tselepidakis, D. P. (1987). "Accommodating the effects high strain rates in modeling the pressure-strain correlation, Rep TFD/87/5, UMIST, Manchester, Dept. Mech. Engrg.

Gibson, M. M. , and Launder , B. E.(1978), Ground Effects on pressure Fluctuations in the Atmospheric Boundary Layers, J. Fluid Mech. , 86, 491-551.

Goncalves E. and R. Houdeville (2001), Reassesment of the Wall Function Approach for RANS Computations, Aerosp. Sci. Technol 5, 1-14.

Gonzalez, J. A. (2005), Ratings for the Pressurized Flow in Gated Culverts with Weir Box Inlet G304A-J and G306A-J,Technical Publication SHDM Report # 2005-02.

Hanjalic, K. , Launder, B. E. (1972) , A Reynold's Stress Model of Turbulence and its Applications to thin shear. Part 4, J. Fluid Mech., 86, 491-551.

Jaw, S.Y, and C. H. Chen (1998), Present Status of Second Order Closure Turbulence Models. I:Overview, J. Engrg. Mech 124(5), 485-501.

Keylock, C. J., Hardy, R. J., Parsons, D. R., Ferguson, R. I., Lane, S. N. and Richards, K. S., 2005. The theoretical foundation and potential for large-eddy simulation (LES) in fluvial geomorphic and sedimentological research, *Earth-Science Reviews.,* 71, 271-304.

Launder, B. E. (1976) , Low Reynol's Number Turbulence turbulence near walls, Rep. TFD/86/4, UMIST, Manchester, Dept. Mech. Engrg.

Launder, B. E. , Morse, A.P. (1979), Numerical Prediction of axi-symmetric free shear flows with a Reynold's Stress Closure, turbulence Shear Flows, Springer , Heidelberg, Germany, 274-295.

MacWilliams, M.L., R. L. Street, and P. K. Kitanidis , Modeling Floodplain Flow on Lower Deer Creek, CA., River Flow 2004: Proceedings of the Second International Conference on Fluvial Hydraulics, Greco, Carravetta, & Della Morte (eds.), Vol. 2, 1429-1439, Balkema, 2004.

McMillan, O. J. , Ferziger, J. H. (1980), Tests of New Subgrid scale Models in Strained Turbulence. AIAA J. 80, 1339.

Menter, F. R., Two Equation Eddy Viscosity Turbulence Modelsfor Engineering Applications, AIAA J.32, 1598-1605.

Nisizima, S. ,and Yoshizawa, A (1987), Turbulent Channel and Couette Flows using an anisotropic k-ε model, AIAA J., 25(3), 414-420.

Normann, J. M., Houghtalen, R. J., and Johnson, W. J. (1985), Hydraulic design of highway culverts. *Hydraulic Design Series No. 5*, 2nd Ed., Federal Highway Administration, Washington, D.C.

Patel V.C, W Rodi and, G Scheuerer (1985) turbulence models for near wall and low Reynold's Number Flows- A review, AIAA J.23 (9): 1308-1319.

Pope, S. B., 2000. Turbulent Flows, Cambridge University Press.

Rodi, W., Ferziger, J. H., Breuer, M., Pourquie, M., 1997. Status of large eddy simulation: reulsts of a workshop, ASME*, J. Fluids. Eng*., 119, 248-262.

Rotta, J. (1951), Sattistical Theory of non-homogeneous Turbulence, Rep. TWF/TN/38, TWF/TN/39, Imperial Coll., London.

Speziale, C. G. , Gatski T. B., and Mac Giolla Mhuiris, N. (1989), A Critical Comparison of Turbulence models for Homogeneous Shear Flows in a Rotating Frame. Proc., 7th Symp Turb. Shear Flows, Standford, Calif., 227.3.1-27.3.6.

Wilcox, D. C., 1988. Turbulence modeling for CFD, DCW Industries

Yakhot, V., and Orszag, S.A. (1986), Renormalization Group Analysis of Turbulence. I. Basic Theory, J. Scientific Computing 1(1), 3-51.

Yen, B.C. (1992). Dimensionally Homogeneous Manning's Formula, *J. Hydr. Eng. Div.*, ASCE, 118(9), 1326–133.