Three-dimensional numerical investigation of flow at 90° open channel junction

التحري العددي ثلاثي الابعاد لملتقى جريان ° ٩٠ في القنوات المفتوحة

Assist. Lecture –Waqid Hameed Al-Mussawi
Civil Engineering Department
Karballa University
Email:Waqed2005@yahoo.com

Lecture Dr. –Musa Habib Al-Shammary
Civil Engineering Department
Karballa University
Email:Mussahabib@yahoo.com

lecture Dr. –Husam Hadi Alwan
Technical Institute
Karballa
Email:Husam44a@yahoo.com

Abstract

Open channel junction flow is of interest in environmental and hydraulic engineering. It occurs in many hydraulic structures such as wastewater treatment facilities, irrigation ditch, fish passage conveyance structures, and natural river channels. This paper provides the details of application of numerical solution (Finite Volume) by FLUENT-3D software in simulation of 90° open channel junction flows. A three-dimensional turbulent model is developed to investigate the flow characteristics (velocity profiles) in open channel junction with different discharge ratios. Were comparisons of Weber et.al.(2001) made between numerical results and measured experimental velocities. Good agreement is obtained between the model simulation and experimental measurements. Statistical analysis of results obtained was, also, applied by using chi-square test (goodness-of-fit test) to ensure the previous conclusion.The correlation coefficient for all tests was not less than (0.952) .

الخلاصــة //

ان الجريان في ملتقيات القنوات المفتوحة يحضى بالاهتمام في مجالي الهندسة البيئية والهايدروليكية.حيث انه يحدث في العديد من المنشأت الهايدروليكية كما في وسائل معالجة مياه الفضلات وقنوات الري ومنشأت تحويل مرور الاسماك في السدود وعند روافد الانهار الطبيعية.ان هذا البحث يقدم تطبيق تفصيلي للحل العددي ثلاثي الابعاد بطريقة الحجوم المحددة(Finite Volume) بواسطة برنامج (FLUENT) لمحاكاة الجريان في منطقة ألتقاء القنوات المفتوحة وبزاوية 90º.استخدم الموديل ثلاثي الابعاد للجريان الضطرب للتحري عن خصائص الجريان (مخططات السرعة) في ملتقيات القنوات المفتوحة ولنسب تصريف مختلفة. تم مقارنة نتائج مخططات السرعة اللابعدية من الموديل العددي مع النتائج المختبرية ل(2001) Weber. ولقد اظهرت المقارنة قبولا جيداً بين النتائج المحسوبة من الموديل والنتائج المختبرية،حيث استخدم أختبار مربع كاي الاحصائي (جودة الموافقة) للتأكد من قبول النتائج المحسوبة،أضافة الى ان معامل الارتباط بين النتائج المحسوبة والنتائج المختبرية لم يقل عن (٠.٩٥٢) .

Introduction

Open channel networks are often encountered in water resources engineering. Typical examples include conveyance structures in urban water treatment plants, irrigation and drainage canals, and natural river system. It also generated theoretical interest due to a number of important flow phenomena involved. Since we are facing different hydraulic conditions to cause changes in channel alignment, combining of two flows accompanies many complex problems such as local sedimentary, channel scour and sidewall erosion and others.

The difficulty in addressing the problem theoretically is that there are numerous factors that influence flow characteristics at the junction of two open channels. One set of variables can be described as geometry variables, such as the size, shape, slope, and angle between the combining channels. Many combinations of these four variables are possible. A second set are flow variables, such as the Froude number in the downstream flow, the channel roughness, the ratio of discharge between the two tributary channels, and the variation of liquid properties. It is readily apparent that a simplified mathematical model is incapable of fully describing the complex flow conditions present at a junction. The difficulty of adequately describing this flow with simplified mathematical models leads to the possibility of using a 3D computational fluid dynamics (CFD) code to describe the flow conditions in a combining open channel junction. Due to a number of important flow phenomena involved, many studies have been conducted to seek the detailed hydrodynamics characteristic of complex junction flow.

Literature review

Taylor (1944) {quted by (Chong, 2006)} first addressed the topic of open-channel junction flow by focusing on the depth ratio between the upstream branches and the downstream channel. The results are a momentum analysis that yields a predictive equation for the depth ratio. Taylor's paper is important for its identification of the need of theoretical description of the open-channel junction and the groundwork it formed for future investigations.

Webber and Greated (1966) began the focus on the general flow characteristics at an open-channel junction. Webber and Greated (1966) implemented the method of conformal mapping to define a theoretical flow pattern throughout the junction region.

Best and Reid (1984) show that a well-defined relationship between the dimensions of the separation which form immediately downstream of the junction and the ratio of discharge from the tributary to the total discharge. Ramamurthy et al. (1988) and Hager (1989) reported extensive studies and methods for the prediction of the depth increase in a transitional flow junction. Kumar (1993) described a refinement and simplification of the predictive equation by Hager.Weerakoon et al. (1991) examined the three-dimensional flow structure at a junction by means of experimental measurements and a computational model incorporating the k-ε turbulence closure scheme.

Weber et al. (2001) performed an extensive experimental stud of combining flows in 90^{o} open channel for the purpose of providing a very broad data set comprising three velocity components, turbulence stresses, and water surface mappings.

Huang et al. (2002) provided a comprehensive numerical study of combining flows in open channel junction and investigate the effect of the junction angle on the flow characteristics .Mao and Wu Rong (2003) provided a numerical simulation of open channel flow in 90^{o} combining junction using Hanjalic-Launder (H-L) modification model to analyze the relative importance of various factors and was compared with laboratory measurements.

A numerical model using finite-element methods was presented by Chong (2006), analysis the open channel junction and a simple comparison with experimental data for velocity profiles were done. Pirzadeh and Shamloo (2007) provide application of FLUENT -2D&3D software in simulation of lateral intake flows. Comparisons have been made between numerical results and measured experimental velocities for a lateral intake. Shamloo and Pirzadeh (2007) provided the application of FLUENT-2D software in simulation of lateral intake flows, using standard *K*-*ε* turbulence and Reynolds-stress (RSM) turbulent models.

The present work

In the current study, an attempt has been made to model fluid flow through open channel T-junction using FLUENT software and numerical 3-D results for velocity profiles with different discharge ratios. The model validation was made by comparing the numerical results with the laboratory experiment results performed by Weber et al. (2001).

Importance of study

The river confluence, which can be defined as a junction of two or more streams, is a frequently found element along a river path. It facilitates to join all streams from many sources over the basin and allows draining them in the form of a river. The river bank erosion and the plunge of the bank due to the lateral flow are critical due to the bank safety. This is well evidenced by the failure of levees at the confluences of many rivers during the floods which resulted serious damages in many countries.

In the design of flood-control channels and fish passage conveyance structures, one of the more important hydraulic problems is the analysis of the flow conditions at open channel junctions. It has recently become more popular to improve the design process using the result from a numerical model study. Applying computational fluid dynamics (CFD) in this field of engineering can improve the final design and accelerate the design process. Also it is importance lies in the fact that the analysis of the junctions is of considerable use to engineers engaged on the design of irrigation systems, drainage systems, channel in sewerage treatment works and other similar projects.

The objective of the present study is the use of 3-D numerical model to predict the velocity profile in the combining flows of open channel junction.

Experimental investigation

Velocity profiles obtained from the current numerical model were compared with laboratory experiment results performed by Weber et al. (2001); in their experimental set-up, the main channel was (21.95 m) long and the branch channel was (3.66 m) long, fitted at (5.49 m)from the upstream main channel . The width of both channels (w) was (0.91 m), the bed slope was zero, and channels were 0.51m deep,as illustrated in (Fig. 1).The upstream main channel, branch channel, and combined tail water flow are denoted as *Qm*, *Qb* and *Qt*, respectively.

The coordinate system defined for the testing had the positive *x*-axis oriented in the upstream direction of the main channel. The positive *y*-direction points to the main channel wall opposite of the channel junction. Thus the positive *z*-axis is upward in the vertical direction. The origin from which all points are measured was the bed at the upstream corner of the channel junction. All distances were normalized by the channel width, *w* = 0.914 m. The nondimensionalized coordinates are called x^*, y^*, and z^* for x/w, y/w, and z/w, respectively. All test sections in this study were denoted by the distance in channel widths measured positive in the *x*-direction for upstream main channel measurements, negative in the *x*-direction for combined tailwater flow measurements, or negative in the *y*-direction for measurements in the branch channel. The velocity measurements have been nondimensionalized by the downstream average velocity, *Vt* = 0.628 m/s.

Flow ratio q^* is defined as the ratio of the upstream main channel flow *Qm* to the constant total flow *Qt*,which is equal to 0.170m^3/s, and the tailwater depth, 0.296 m, were held constant. This constant downstream flow rate and depth produced a constant tailwater Froude number, F*t* = 0.37, and a constant tailwater average velocity, *Vt* = 0.628 m/s. The flow conditions tested are listed in Table 1. In the experiment, a total of six runs were conducted. In this study, only q* equal to 0.420 and 0.750 are chosen, which represent two distinct flow scenarios with quite different flow discharges.Only were selected for model simulation.

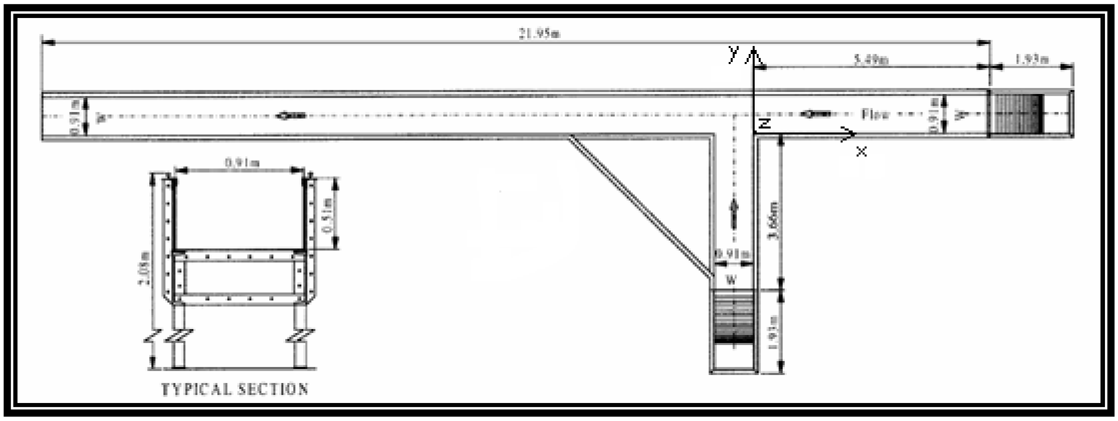

Fig.1. Layout of experimental flume.

Table1: Flows tested

Qm (m^3/s)	*Qb* (m^3/s)	*Qt* (m^3/s)	*q*=Qm/Qt*
0.014	0.156	0.170	0.083
0.042	0.127	0.170	0.250
0.071	0.099	0.170	0.420
0.099	0.071	0.170	0.583
0.127	0.042	0.170	0.750
0.156	0.014	0.170	0.920

Numerical model description

FLUENT is the CFD solver for choice for complex flow ranging from incompressible (transonic) to highly compressible (supersonic and hypersonic) flows. Providing multiple choices of solver option, combined with a convergence-enhancing multi-grid method, FLUENT delivers optimum solution efficiency and accuracy for a wide range of speed regimes. The wealth of physical models in FLUENT allows you to accurately predict laminar and turbulent flows, various modes of heat transfer, chemical reactions, multiphase flows, and other phenomena with complete mesh flexibility and solution- based mesh adoption (Fluent user guide 2003).

FLUENT solves governing equations sequentially using the control volume method. The governing equations are integrated over each control volume to construct discrete algebraic equations for dependent variables. These discrete equations are linearized using an implicit method. As the governing equations are nonlinear and coupled, iterations are needed to achieve a converged solution.
Conservative form of the Navier-stokes equations using the finite volume method on structured orthogonal, Cartesian coordinate's grid system.
Turbulent flows can be simulated in FLUENT using the standard k-ε, large eddy simulation (LES), renormalization group (RNG), or the Reynolds-stress (RSM) closure schemes.
The model optimizes computational efficiency by allowing the user to choose between various spatial (Second-order upwind, third-order, QUICK) discritization scheme. The under relaxation factors are chosen between 0.2 and 0.5.
Turbulent stresses in Reynolds-averaged equations can be closed using any of several exiting turbulence models. No single turbulence model is accepted universally for solving all closes of problems but each model has certain advantages over the others depending on the type and the nature of the flow field to be simulated and the desired accuracy of results (Neary and Odgaard, 1993).
The simplest and most widely used two-equation turbulence model is (k-ε) model that solves two separate equations to allow the turbulent kinetic energy and dissipation rate to be independently determined (Ferzieger and Peric, 1996).

The turbulence kinetic energy, k, is modeled as:

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\nu_T}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + P_k - \varepsilon \qquad (1)$$

Where P_k is given by:

$$P_k = \nu_T \frac{\partial U_j}{\partial x_i}\left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j} \right) \qquad (2)$$

$$\nu_T = c_\mu \frac{k}{\varepsilon^2} \qquad (3)$$

The dissipation of k is denoted ε, and modeled as:

$$\frac{\partial \varepsilon}{\partial t} + U_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\nu_T}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) + C_{\varepsilon 1} \frac{\varepsilon}{k} p_k + C_{\varepsilon 2} \frac{\varepsilon^2}{k} \qquad (4)$$

Typical values of the empirical coefficients in the k-ε model (Rodi, 1980):
c_μ=0.09, $C_{\varepsilon 1}$=1.44, $C_{\varepsilon 2}$=1.92, σ_k=1.0 and σ_ε=1.30

We used second order upwind discritization scheme for Momentum, turbulent kinetic energy and turbulent dissipation rate; used body force weighted discritization scheme for Pressure and SIMPLE algorithm for Pressure-Velocity Coupling Method. Also, the standard (k-ε) model is used in the present study to simulate the effect of turbulence.

Governing Equations

The governing equations of fluid flow in rivers and channels are generally based on three-dimensional Reynoldes averaged equations for incompressible free surface unsteady turbulent flows as follows (Neary and Odgaard, 1993):

$$\frac{\partial U_i}{\partial t}+U_j\frac{\partial U_i}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\left(-P+\frac{2}{3}k\right)\delta_{ij}+v_T\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)\right] \qquad (5)$$

Where:

t = time ;Ui(i=1,2,3) and u_i(i=1,2,3)= are the mean and fluctuating velocity components in the x (lateral), y (streamwise) and z (vertical) direction, respectively ; *P*=total pressure; *k*=turbulently kinetic energy; δ_{ij}=Kronecker delta; v_T=turbulent viscosity; and *j*=1, 2, 3.

There are basically five terms: a transient term and a convective term on the left side of the equation. On the right side of the equation there is a pressure/kinetic term, a diffusive term and a stress term.

In the current study, it is assumed that the density of water is constant through the computational domain. The governing differential equations of mass and momentum balance for unsteady free surface flow can be expressed as (Chen and Lian, 1992):

$$\frac{\partial u_i}{\partial x_i}=0 \qquad (6)$$

$$\frac{\partial u_i}{\partial t}+u_j\frac{\partial u_i}{\partial x_j}=-\frac{\partial P}{\partial x_i}+g_{xi}+v\nabla^2 u_i \qquad (7)$$

Where *v* is the molecular viscosity; g_{xi} is the gravitational acceleration in the x_i direction.

As in the current study, only the steady state condition has been considered, therefore equation (6) and (7) incorporate appropriate initial and boundary conditions deployed to achieve equilibrium conditions.

Boundary and Initial Conditions

Appropriate conditions must be specified at domain boundaries depending on the nature of the flow. In simulation performed in the present study, velocity inlet boundary condition used for two inlets (main cannel and branch) at all of runs. Outlet pressure is used to define outlet boundary conditions. The length of the main and branch channels were chosen properly with experimental dimensions (Fig.1). The no-slip boundary condition is specified to set the velocity to be zero at the solid boundaries and assumed to be smooth. In simulation performed in the first case of present study, velocity inlet boundary condition is specified, and calculated depending on the discharge ratio. Two discharge ratios *q*=Qm/Qt* equal to *0.42* and *0.75* were used corresponding to the Weber et al. experiments.

A finite volume approach is used for the solution of the governing equations. Second order upwind scheme is selected to discretize the governing equations. SIMPLE algorithm is used for pressure-velocity coupling.

It is also important to establish that grid independent results have been obtained. The grid structure must be fine enough especially near the wall boundaries and the junction, which is the region of rapid variation. Various flow computational trials have been carried out with different number of grids in x and y directions. The mesh with (2D quadrilateral, 3D hexahedral.) cells, created by using GAMBIT model as shown in Fig. (2), is used in this study. This mesh then would be exported to FLUENT in order to be used in the solution.
In three dimensions simulations performed in the present study, two separate inlets for air and water are specified. At each inlet, uniform distributions are given for all of dependent variables. Two separate outflows for air and water are specified at the end. At the top surface above the air, zero normal velocity and zero normal gradients of all variables are applied by defining a symmetric boundary condition.

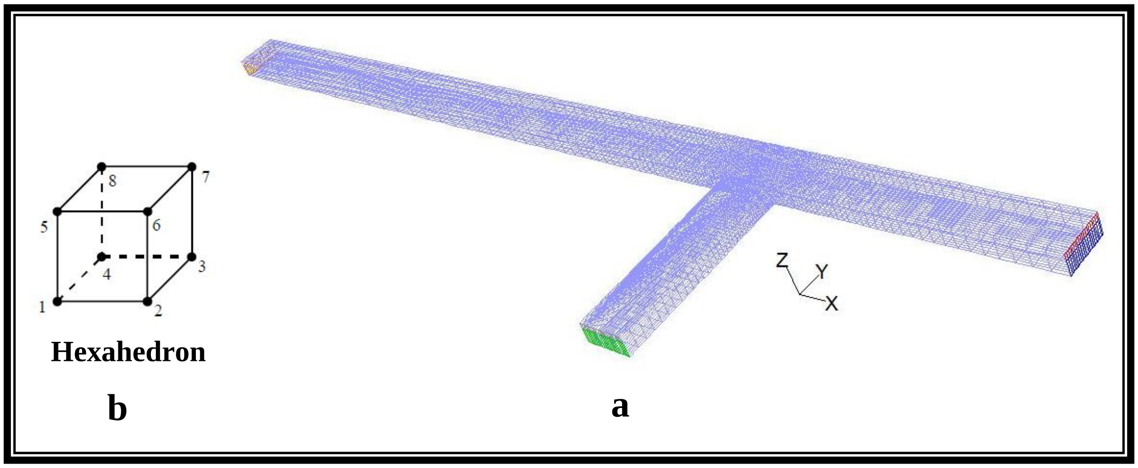

Fig.2. Three dimensional computational geometry and grid (a), the type of volume element (b)

Validation model

The three dimensional (CFD) model described above is validated first by calculation two flow scenarios of the experiment of Weber et al. (2001) with the 90° junction flow. Then, Statistical analysis of the results obtained was, also, applied by using the chi-square test (goodness-of-fit tests) to ensure acceptably of the model.

The chi-square test is the more interesting statistical tests in the engineering field so that it can be used in this analysis. The equation of chi-square is (William, 1980):

$$\chi^2 = \sum_{j=1}^{r} \frac{\left(fo_j - fe_j\right)}{fe_j} \qquad (8)$$

Where: fo_j : observed values, fe_j : expected values and r: number of rows.

Moreover, the correlation coefficient will be obtained between the calculated and observed velocities.

Results and Discussions

In this paper numerical investigation are performed for evaluation of the ability of an available 2D & 3D flow solver to cop with the fully turbulent flow in 90° junction. The velocity profiles of turbulent junction open channel flow are of great interest to engineers, particularly in determining the surface resistance to flow which has practical consequences, such as in the estimation of soil erosion and sediment transport in alluvial channels.

In Figures (3) to (7) and (8) to (12) results of the numerical model are compared with experimental X- velocity profiles in the main channel at 5 non-dimensional locations at x-direction for discharge ratios (*q*=0.750* and *0.420*) respectively. The velocities are non-dimensionalized with respect to the average downstream, tail water velocity (0.628 m/s).From these figures, it can be concluded that two dimensional results generally have reasonable agreement with measured ones, but at some sections the computed results do not agree very well with those measured, which might be partly due to the three dimensional effects. Further, Weber et al. (2001) presented flow velocities at *z**=0.278 but in the current 2D-study surface velocities have been used. Moreover, due to limitation of two-dimensional numerical model, the vertical acceleration is neglected in the current 2D-study.Both results from experimental and numerical models give the negative X-velocities in figures (4, 5, 6,9,10, and 11), these values showed that the separation zone (recirculation) was observed immediately downstream of the junction due to deflection from outer wall.

The application of the results of chi-square equation on the velocities data from 2D &3D using level of significance ($\alpha = 0.05$ and $\alpha = 0.1$) are shown in Table (2). It is shown from these analysis, that the chi-square values (χ^2) from equation for all cases less than the chi-square values (χ^2_α) tabulated for level of significance ($\alpha = 0.05$ and $\alpha = 0.1$). From this tabil it can be seen that the results of 3D-model are near to the observed velocities from the results of 2D-model at the same point.Also,it can be shown that the resuls of 3D model are at least error,which results from the 2D model.

3D-results of numerical modeling represent a better agreement than results of two dimensional modeling indicating the effect of turbulence in the region with three dimensional complex features.

Fig. (13) & (14) displays X-velocity contours near the water surface for discharge ratios (*q*=0.75* and *0.25*) respectively. Note that there are blank spots for the experimental plots as measured data are not available in those areas. The separation zone can be seen as the area of low velocity along the junction adjacent wall immediately downstream of the channel junction. Recirculation inside the separation zone is shown as the region of positive velocity, indicating upstream motion. The largest velocities occur just downstream of the junction at *x** = -2.0, in the main channel flow region contracted by the zone of separation. Both results show the increasing of velocity at outer wall region downstream of the junction.

For completeness of the comparison, Figures (15) to(19) and (20) to(24) show the comparison of stremwise velocity component (it is *u* in the main channel) near the outer wall region of the junction (*y**=0.875) where the velocities were higher at the same selected stations in x-direction for discharge ratios (*q*=0.75* and *0.42*) respectively. It is seen that a good agreement between the prediction and measurements is obtained. The results indicate that the computed stremwise velocity showed trends that are similar to observations made by Weber et al. (2001).

Table2: Chi-square test for 2D and 3D velocities results

q*	X*	Correlation coefficient		(χ^2) calculated		(χ^2_α) Tabulated $\alpha = 0.05$	(χ^2_α) Tabulated $\alpha = 0.1$
		2D	3D	2D	3D		
0.42	-1.00	0.926	0.952	0.552	0.120	15.507	13.361
	-1.33	0.970	0.986	1.458	1.321	15.507	13.361
	-1.66	0.979	0.991	4.330	2.370	15.507	13.361
	-2.00	0.983	0.996	3.022	0.745	12.591	10.644
	-3.00	0.994	0.995	2.247	1.401	12.591	10.644
0.75	-1.00	0.959	0.987	1.397	0.474	15.507	13.361
	-1.33	0.967	0.990	1.378	0.247	15.507	13.361
	-1.66	0.983	0.991	1.000	0.720	15.507	13.361
	-2.00	0.994	0.995	0.633	0.574	12.591	10.644
	-3.00	0.982	0.987	1.250	0.594	12.591	10.644

Conclusions

The two and three-dimensional CFD modeling is used to solve a study state incompressible Reynolds-averaged Navier Stokes equations with (k-ε) turbulence closure model which is called as FLUENT program. The Reynolds-averaged Naver Stokes equations are discritized using a control volume method and solved using the SIMPLE algorithm. Iterations proceed are continued until the sum of the normalized residuals is smaller than 10^{-3}.

A numerical model results are employed to estimate the velocities profiles for downstream main channel in 90° junction with the fully turbulent flow.comparisons have been made between numerical results and measured experimental velocities with different discharge ratio.comparisons indicate that the 2D simulating captures most experimental trendssonable agreement.The study demonstrates that a robust 3D model can effectively supplement experimental studies in understanding the complex flow field in 90° open channel junction.This means that the assumptions used for this model are approximately correct and give some reliability to use this model in other applications.

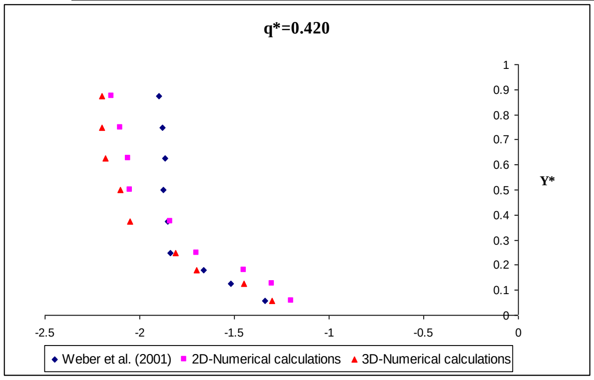

Fig.3.X-Velocity profile in the main channel (X*=-1)

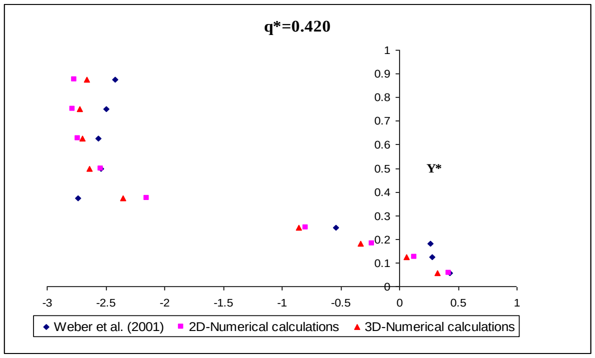

Fig.4.X-Velocity profile in the main channel (X*=-1.33)

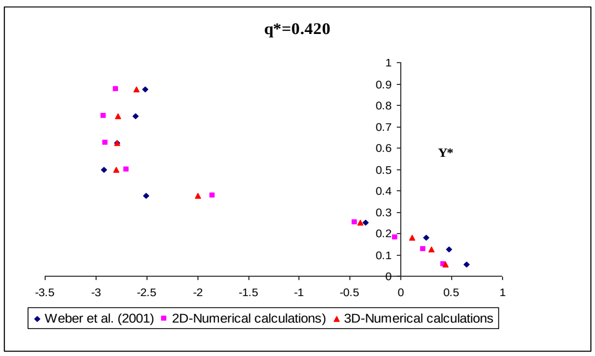

Fig.5.X-Velocity profile in the main channel (X*=-1.66)

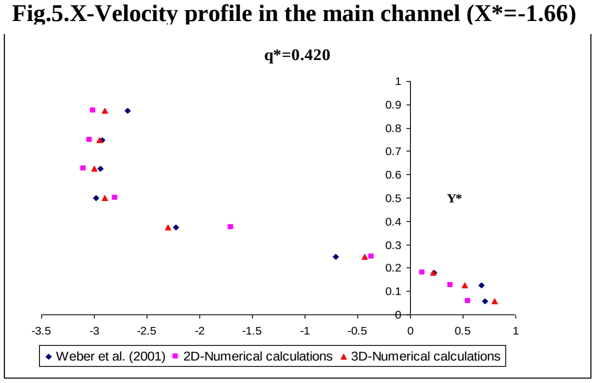

Fig.6.X-Velocity profile in the main channel (X*=-2)

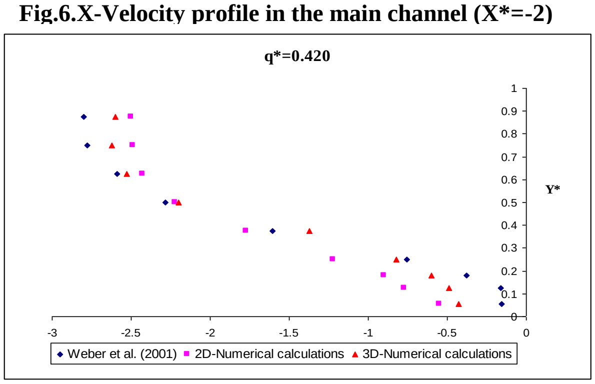

Fig.7.X-Velocity profile in the main channel (X*=-3)

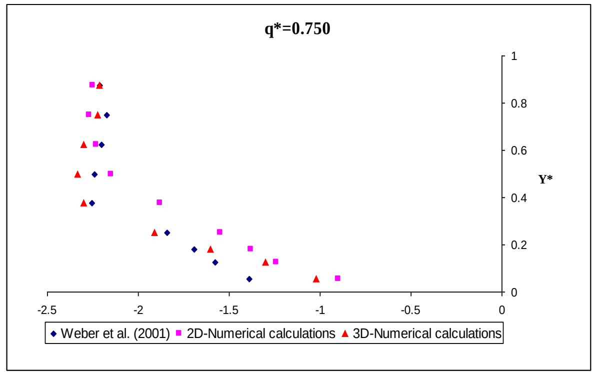

Fig.8.X-Velocity profile in the main channel (X*=-1)

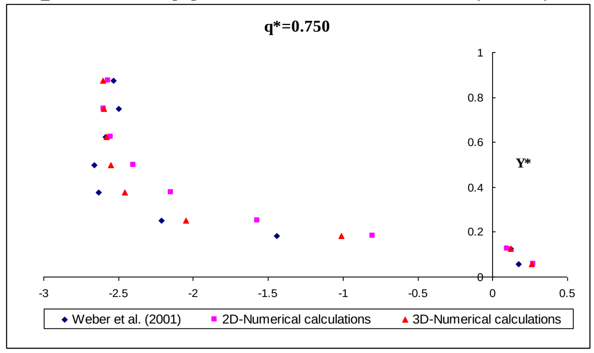

Fig.9.X-Velocity profile in the main channel (X*=-1.33)

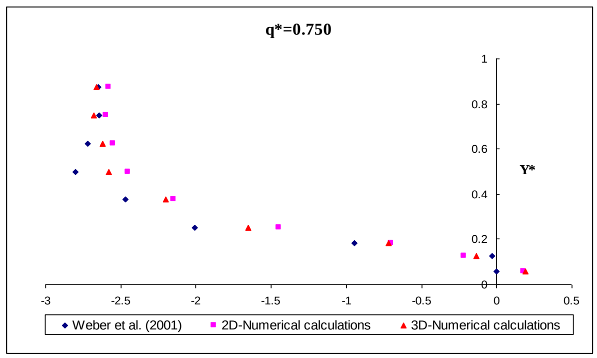

Fig.10.X-Velocity profile in the main channel (X*=-1.66)

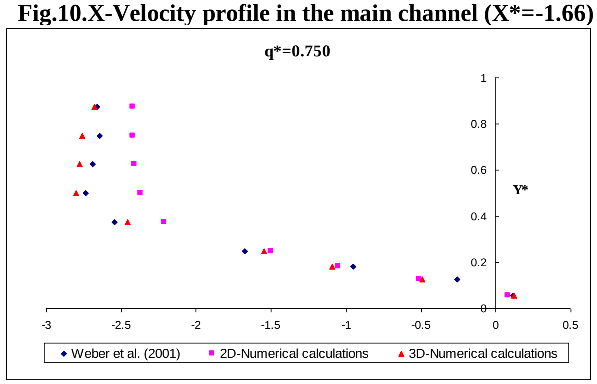

Fig.11.X-Velocity profile in the main channel (X*=-2)

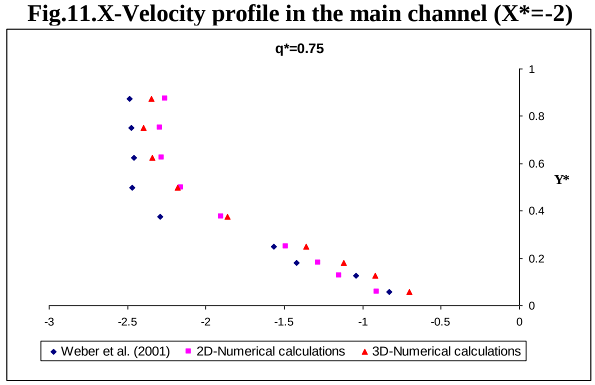

Fig.12.X-Velocity profile in the main channel (X*=-3)

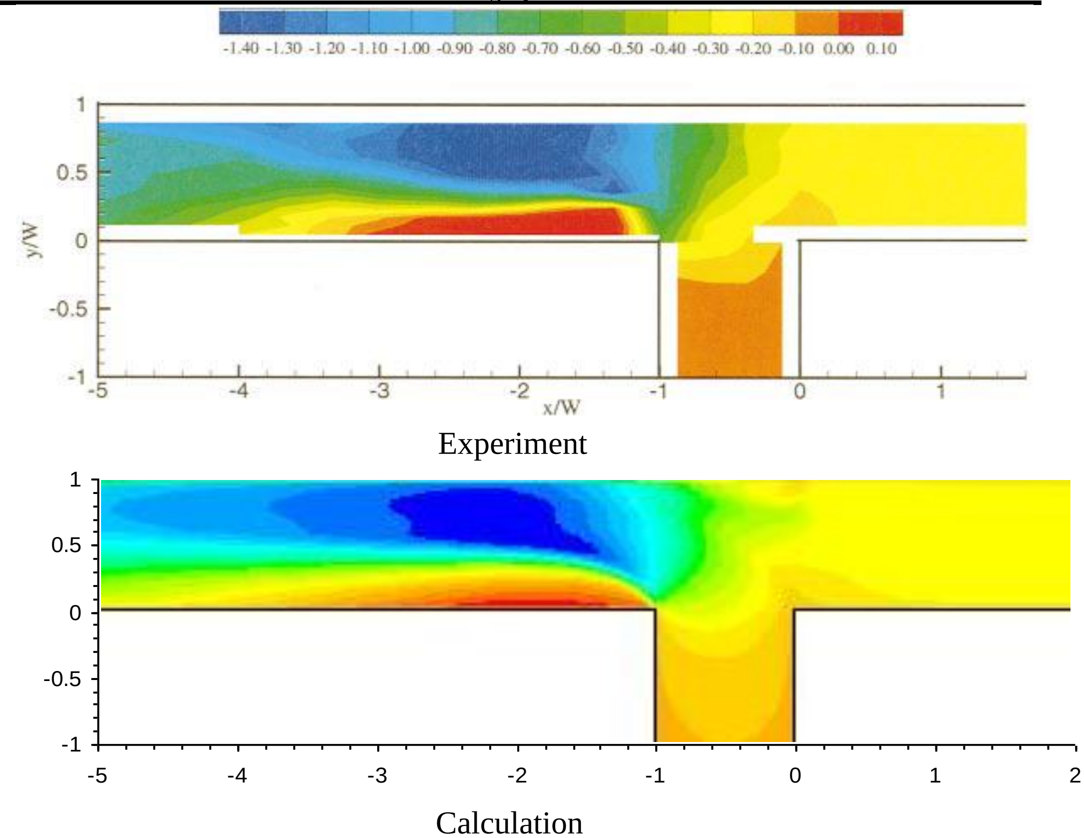

Fig.13.Experimental and numerical comparison of X-velocity distribution at Z*=0.278 for q*=0.750

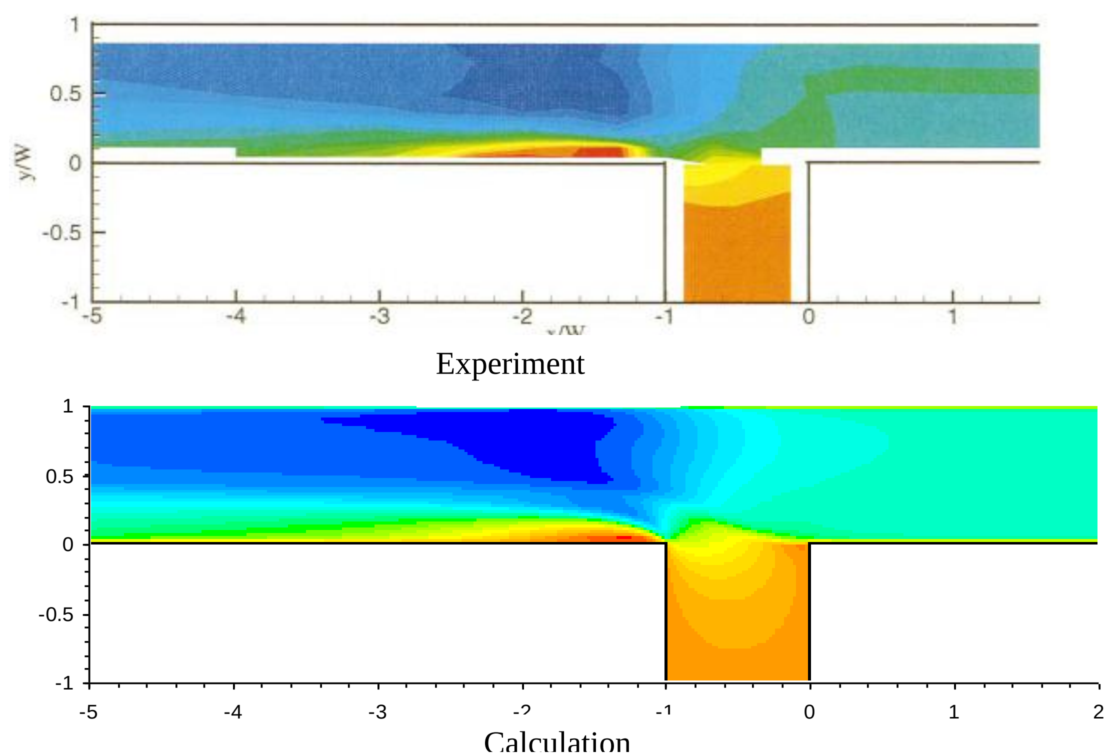

Fig.14.Experimental and numerical comparison of X-velocity distribution at Z*=0.278 for q*=0.250

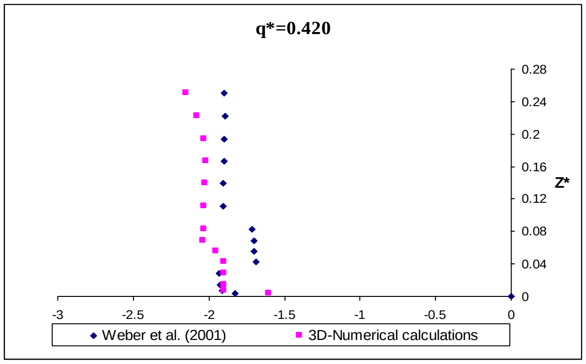

Fig.15.Stremwise velocity profile in the main channel (X*=-1)

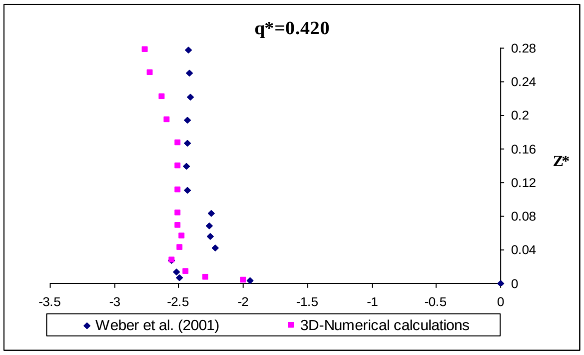

Fig.16.Stremwise velocity profile in the main channel (X*=-1.33)

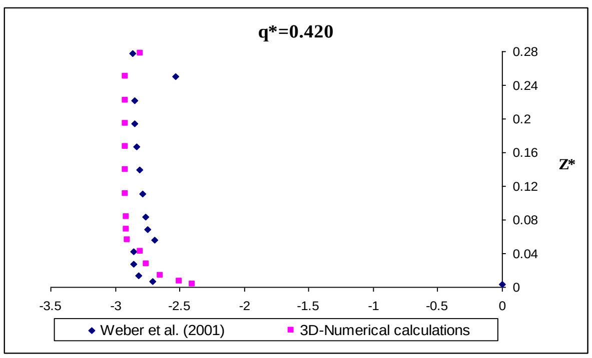

Fig.17.Stremwise velocity profile in the main channel (X*=-1.66)

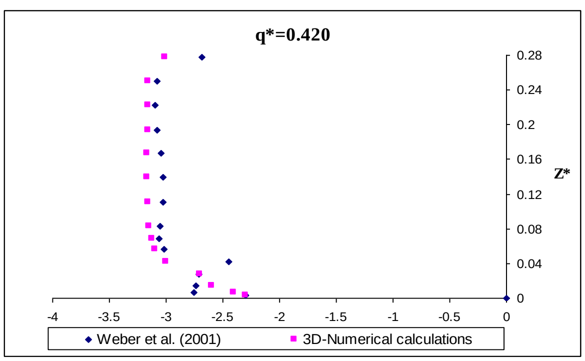

Fig.18.Stremwise velocity profile in the main channel (X*=-2)

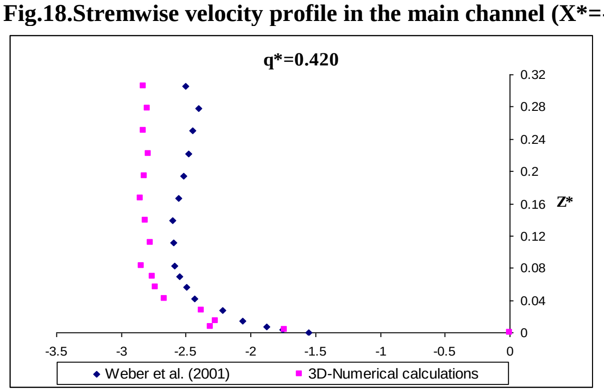

Fig.19.Stremwise velocity profile in the main channel (X*=-3)

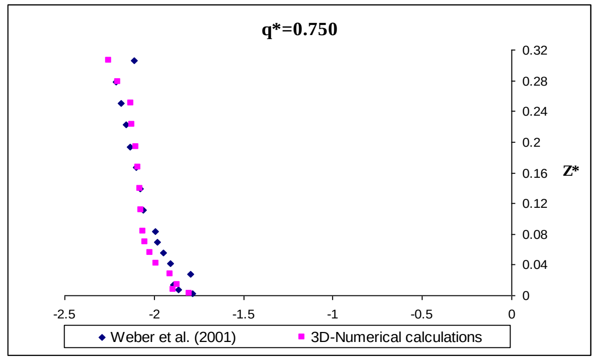

Fig.20.Stremwise velocity profile in the main channel (X*=-1)

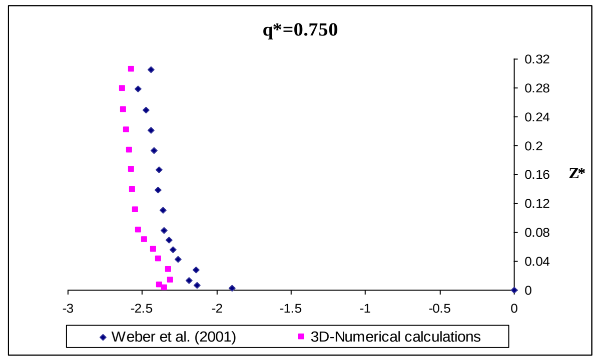

Fig.21.Stremwise velocity profile in the main channel (X*=-1.33)

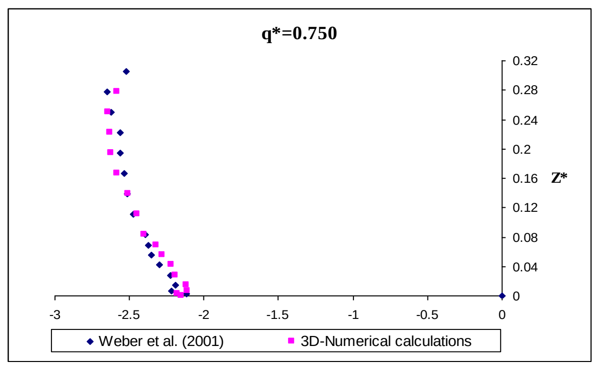

Fig.22.Stremwise velocity profile in the main channel (X*=-1.66)

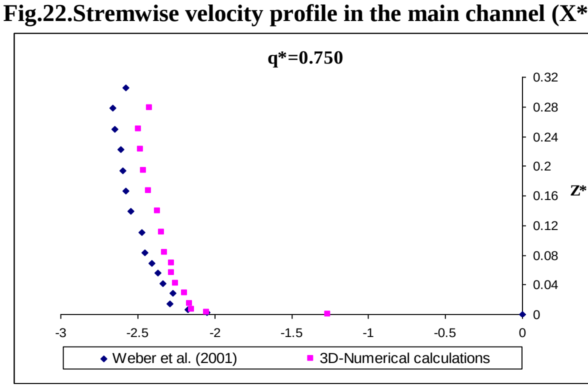

Fig.23.Stremwise velocity profile in the main channel (X*=-2)

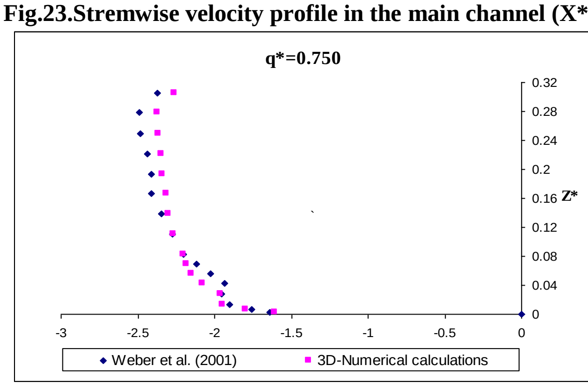

Fig.24.Stremwise velocity profile in the main channel (X*=-3)

References

- Best, J. L. and Reid, I. (1984), "Separation zone at open channel junctions." Journal of Hydraulic Engineering. ASCE, Vol. 110, no. 11, pp.1588-1594.
- Chen H. and Lian G. (1992),"The numerical computation of turbulent flow in T-junctions.", Journal of Hydrodynamics, Series B, No.3, pp.19-25.
- Chong, N.B. (2006), "Numerical simulation of supercritical flow in open channel." M.S.C thesis, University Technology Malaysia.
- Ferzieger, J.L., and Peric, M. (1996),"Computational methods for fluid dynamics." Springer-Verlage, Heidelberg.
- FLUENT user's guide manual-version 6.1., Fluent Incorporated, Lebanon, N.H., 2003.
- Hager, W.H. (1989),"Transitional flow in channel junctions." Journal of Hydraulic Engineering, ASCE, Vol. 115, no.2, pp.243-259.
- Huang, J., Weber, L.J., Lai, Y.G. (2002), "Three dimensional numerical study of flows in open channel junctions.", Journal of Hydraulic Engineering. ASCE, Vol.128, no.3, pp.268-280.
- Kumar, S.G. (1993),"Transitional flow in channel junctions." Journal of Hydraulic Research, Delft, The Netherlands, Vol.31.no.5, pp.601-604.
- Mao, Z. and Rong, W. (2003), "Numerical simulation of open-channel flow in 90-degree combining junction.", Journal of Tsinchua Science and Technology, Vol.8, no.6, pp.713-718.
- Neary, V.S. and Odgaard, A.J. (1993), "Three dimensional flow structure at open-channel diversions. ", Journal of Hydraulic Engineering. ASCE, Vol.119, no.11, pp.1223- 1230.
- Pirzadeh, B. and Shamloo, H. (2007), "Numerical investigation of velocity field in dividing open channel flow." Proceedings of the 12th WSEAS International Conference on applied mathematics, Cairo, Egypt, Desember29-31, 2007, pp.194-198.
- Ramamurthy,A.S.,Carballada,L.B.,and Tran,D.M.(1988),"Combining open-channel flow at right-angled junctions." Journal of Hydraulic Engineering. ASCE, Vol.114, no.12, pp.1449-1460.
- Rodi, W. (1980)," Turbulence models and their application in hydraulics."A state of the art review, IAHR, Delft, The Netherlands.
- Shamloo, H. and Pirzadeh, B. (2007), "Investigation of characteristics of separation zones in T-Junctions.", Proceedings of the 12th WSEAS International Conference on applied mathematics, Cairo, Egypt, Desember29-31, 2007,pp.189-193.
- Taylor, E.H. (1944), "Flow characteristics at rectangular open-channel junctions. ", Transactions, ASCE, Vol.109, pp. 893–902.
- Weber, L. J., Schumate, E. D., and Mawer, N. (2001), " Experiments on flow at a 90◦ open-channel junction.", Journal of Hydraulic Engineering, ASCE, Vol. 127, no. 5, pp. 340-350.
- Webber,N.B.,and Greated,C.A.(1966),"An investigation of flow behavior at the junction of rectangular channels."Proc.Instn.of Civil Engineering, Vol. 34, Thomas Telford Ltd., London, pp.321-334.
- Weerakoon, S. B., Kawahara, Y. and Tamai, N. (1991), "Three-dimensional flow structure in channel confluences of rectangular section.", International Association for Hydraulic Research, A, pp.373-380.
- William, L. Hays, (1980)," Statistics", third edition, P.539.