INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN FLUIDS
Int. J. Numer. Meth. Fluids 2003; **42**:923–928 (DOI: 10.1002/fld.563)

Effects of under-relaxation factors on turbulent flow simulations

R. M. Barron[1,*,†] and Ali A. Salehi Neyshabouri[2]

[1]*Fluid Dynamics Research Institute, University of Windsor, Windsor, Ont., Canada N9B 3P4*
[2]*Civil Engineering, University of Tarbiat Modarres, P.O. Box 14155-4838, Tehran, Iran*

SUMMARY

Under-relaxation factors are significant parameters affecting the convergence of a numerical scheme. Some earlier work has been done to optimize these parameters, but this was restricted to special flow domains, and the range of changes for under-relaxation factors and convective algorithms are limited.

In this paper, the effects of changing under-relaxation factors for different variables, different convective schemes and grid sizes on the convergence of the numerical solution of three 2D turbulent flow situations are studied. These three flows are duct flow, trench flow and inclined free falling jet flow.

KEY WORDS: under-relaxation factors; duct flow; trench flow; inclined free falling jet flow

INTRODUCTION

One set of most significant parameters affecting the convergence of a numerical scheme is the under-relaxation factors. Some earlier work has been done to optimize these parameters. However, these previous works are restricted to special flow domains and the range of changes for under-relaxation factors and convective algorithms are limited, e.g. References [1, 2].

In this paper, the effects of changing under-relaxation factors for different variables, different convective schemes and grid sizes on the convergence of the numerical solution of three 2D turbulent flow situations are studied. These three flows are duct flow, trench flow and inclined free falling jet flow.

*Correspondence to: R. M. Barron, Fluid Dynamics Research Institute, University of Windsor, 401 Sunset Avenue, Windsor, Ont., Canada N9B 3P4.
†E-mail: az3@uwindsor.ca

Received September 2001
Revised 12 October 2002

FLOW EQUATIONS AND NUMERICAL SCHEMES

Steady, turbulent, incompressible fluid flow is governed by equations representing conservation of mass and linear momentum. Turbulence is modelled using the standard $k-\varepsilon$ turbulence model with wall functions. Standard boundary conditions are used at inlets, outlets, walls and free surfaces.

The flow equations are integrated over each cell using a finite-volume technique. Two different convective schemes, namely the power-law scheme (POW) and second-order upwind scheme (SOU), are used to compute the convective fluxes. For velocity–pressure coupling, the SIMPLE algorithm is followed. Since a non-staggered grid is used in this study, the Rhie and Chow [3] interpolation method is used to avoid instabilities in the calculation of the velocities and pressure.

It is necessary to limit the change in each variable between iterations (under-relaxation). It is not possible to precisely analyse the convergence of the numerical method, so the selection of under-relaxation factors is largely empirical [4].

TEST CASES

To determine a general range of acceptable under-relaxation factors for variables involved in the numerical simulation of turbulent flow, three flows are considered.

Duct flow (*Test case T1*)

Numerical simulation of the flow domain between two walls was studied. The aspect ratio of the physical domain was six, with a uniform inlet velocity of 5.0 m/s. Three mesh sizes, 30×10(T1a), 45×15(T1b) and 60×20(T1c), (in x and y directions) were used. For discretization of the convective terms, the POW scheme was adopted.

Trench flow (*Test case T2*)

Experimental results for this flow case have been presented by van Rijn [5] and numerical simulation was done by Basara and Younis [6]. Therefore, we are able to check the validity of the present numerical model, using the location of the point of reattachment for validation. The SOU scheme was applied, with an inlet velocity in x direction of 0.5 m/s, and mesh size of 130×40.

Inclined free falling jet (*Test case T3*)

In this case the top boundary is a free surface, while the left and bottom boundaries are solid walls. The incoming jet had a width of 26.67 cm and impinged on the free surface at an angle of 75°. The POW scheme was implemented on a 60×10 mesh.

ANALYSIS OF THE RESULTS

The under-relaxation factors for velocity (α_u, α_v), pressure (α_p), turbulent kinetic energy (α_k), dissipation of turbulent kinetic energy (α_ε), eddy viscosity (α_ν) and generation term (α_g)

were systematically changed between the limits of 0.1–0.9 and the divergence or the number of iteration (Niter) required to reach convergence was recorded in each case. The requirement for convergence was that the non-dimensional residuals of all variables be less than 0.0001.

Based on test cases considered for the present work and also considering previous work done by Gopinath and Ganesan [2], it was concluded that the under-relaxation factor for velocity components (in this work $\alpha_u = \alpha_v$) have the most significant effect on the convergence rate. Having first determined an optimal range for α_u and α_v, it is then necessary to check for the interactions between any two other factors.

(*a*) *Effects of other factors on the behaviour of* α_u

For $0.1 \leqslant \alpha_p \leqslant 0.2$ there is a smooth decrease of Niter with increase in α_u and the number of iterations at any α_u decreases as α_p increases from 0.1 to 0.2. For $\alpha_p = 0.3$, the same trend as for α_p between 0.1 and 0.2 is observed, except for trench flow and the finest mesh size of duct flow. For $\alpha_p \geqslant 0.4$ there exist either a limited zone of smooth behaviour with a minimum Niter that is high compared to minimum values which could be obtained by α_p between 0.1 and 0.2, or there is complete divergence. Also, with finer meshes the possibility of divergence increases for the same value of α_p above 0.2. One can choose values of $\alpha_u = 0.9$ or, with acceptance of a small increase of Niter, choose $\alpha_u = 0.8$ which gives a wider range of safe α_p values (Figure 1).

For $\alpha_k = 0.1$, both the trench and inclined jet calculations show divergence or slow convergence for all α_u. For α_k between 0.2 and 0.4 there is no divergence in the solution. $\alpha_k = 0.3$ gives the best convergence rate especially for larger values of α_u. Large values of α_k (0.8 and greater) cause divergence for all flow cases considered. The α_k values between 0.5 and 0.8 also cause divergence in some flow cases or for some values of α_u (especially larger α_u values) and should be avoided.

The solutions are well behaved for $\alpha_g = 0.1$ and 0.2, showing a smooth decrease of Niter with increase of α_u. For simpler flow cases and coarser meshes, increase of α_g above 0.2 causes immediate divergence. For trench flow, divergent behaviour occurs for $\alpha_g \geqslant 0.6$. For flow of the inclined jet, divergence occurs for $\alpha_g \geqslant 0.5$.

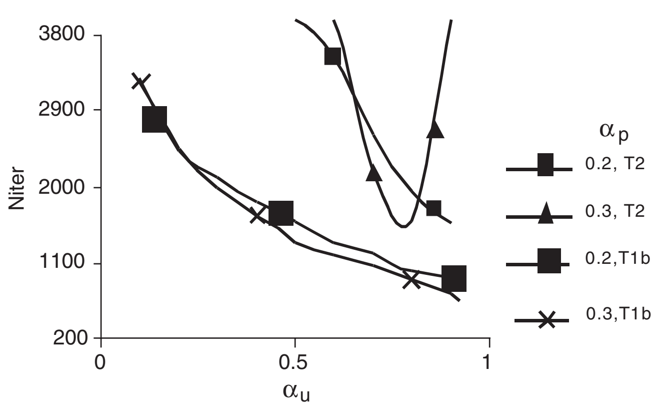

Figure 1. Effect of α_p on α_u (Trench and duct flow).

It was found that α_ε values in the range of 0.1–0.2 cause complete divergence or an oscillating trend between divergence and convergence. For the finer mesh size of duct flow, this behaviour extends up to $\alpha_\varepsilon = 0.4$. Beyond these ranges for α_ε, convergence is obtained. Fastest convergence occurs for $\alpha_\varepsilon = 0.6$.

It seems that values of α_v have the least effect and, for all values of α_v from 0.1 to 0.9, we see a smooth decrease of Niter with increase of α_u. However, the smaller values of α_v (between 0.1 and 0.3) give slightly better convergence speed and $\alpha_v = 0.3$ gives the smoothest curves in most of the flow cases and for different α_u values.

(*b*) *Effects of other factors on* α_p

Increasing the value of α_k in the range of 0.2–0.4 causes the range of safe values for α_p to become narrower in some flow cases and beyond $\alpha_k = 0.4$, divergence occurs. α_g values in the range of 0.1–0.2 do not have much effect on the behaviour of α_p. Higher values of α_g can lead to divergence for any α_p in some cases. In general, increasing the α_ε value above 0.4 increases the range of safe values of α_p, or keeps it the same. Even though α_v does not have much effect on Niter when other factors are kept in their safe ranges, it does have an effect on the range of acceptable α_p values. As α_v increases from 0.1, the extent of safe values of α_p decreases.

(*c*) *Effects of other factors on* α_k

The range of safe values of α_k is reduced when the α_g value is increased to 0.3 in the case of duct flow and to 0.4 for trench flow (Figure 2). For α_g in the recommended range of 0.1–0.2, only in the case of duct flow with the coarsest mesh, there exists a tendency to decrease the limit of safe values of α_k by increasing the α_g value from 0.1 to 0.2. Therefore, it is preferable to choose $\alpha_g = 0.1$. All cases indicate that increasing the α_ε value causes the range of acceptable α_k values to become wider. However, increasing the α_ε value above 0.7 causes the rate of convergence to decrease in the case of trench flow. The only effect of α_v on α_k is to change the range of safe α_k values. Even though this effect is rather negligible,

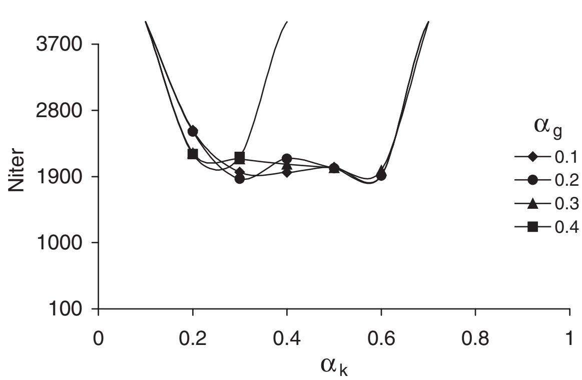

Figure 2. Effect of α_g on α_k (Trench flow).

it seems logical to keep the value of α_v small, preferably below 0.3 to have a wider range of acceptable values of α_k.

(*d*) *Effects of other factors on* α_g

Increasing the value of α_ε causes a slight increase in the range of acceptable α_g values, but does not significantly affect Niter. In the case of duct flow, values of α_v in the low and intermediate ranges do not have any significant effect on α_g. In the case of trench flow, both low and high values of α_v cause divergence for α_g values greater than 0.4. However, for the recommended range of α_g between 0.1 and 0.2, the effect of α_v is negligible.

(*e*) *Effects of other factors on* α_ε

The effects of α_v on the behaviour of α_ε, except for $\alpha_v = 0.7$ in the case of the medium size mesh for duct flow, is negligible. If we choose the value of α_ε in the range of 0.5–0.9, we can neglect any effect of α_v on the α_ε behaviour.

(*f*) *Effects of mesh size*

Three different mesh sizes have been used in the duct flow calculations. From this limited testing, one can observe the effects of mesh size on all the under-relaxation factors. In order to accelerate convergence, as the mesh size increases, α_u should be increased, α_p should be decreased and all other parameters can be kept the same provided they are already in their range of safe values.

CONCLUSIONS

Table I illustrates the conclusions of the present work. This table shows two ranges for any under-relaxation factor, namely the range of safe values, which are based on the non-divergent solution, and recommended range or value, which is narrower than the safe range and results in faster convergence. The range of safe values are applicable even when the mesh size is increased, but increasing α_u and decreasing α_p increases the convergence rate.

An attempt has been made in this work to consider a wider variation of flow cases with two discretization schemes for the convective terms and with different mesh sizes. Consequently, this study has lead to more general conclusions about proper selection of different under-relaxation factors than any previous work.

Table I. Ranges of safe and recommended values for under-relaxation factors.

Under-relaxation factor	α_u, α_v	α_p	α_k	α_g	α_ε	α_v
Range of safe values	0.1–0.9	0.1–0.2	0.2–0.4	0.1–0.2	0.5–0.9	0.1–0.9
Recommended values	0.8–0.9	0.2	0.3	0.1	0.6	0.1–0.3

REFERENCES

1. Peric M, Kessler R, Scheuerer G. Comparison of finite volume numerical methods with staggered and collocated grids. *Computers and Fluids* 1988; **6**(4):389–403.
2. Gopinath R, Ganesan V. Orthogonal arrays: An introduction and their application in optimizing under-relaxation factors in 'SIMPLE'-based algorithm. *International Journal for Numerical Methods in Fluids* 1992; **14**: 665–680.
3. Rhie C, Chow W. Numerical study of the turbulent flow past an airfoil with trailing edge separation. *AIAA Journal* 1983; **21**:1525–1532.
4. Ferziger J, Peric M. *Computational Methods for Fluid Dynamics*. Springer: Berlin, Heidelberg, 1997.
5. van Rijn L. The computation of the flow and turbulence field in dredged trenches. *Report S*-488-I, Delft Hydraulics Laboratory, Delft, The Netherlands, 1982.
6. Basara B, Younis B. Prediction of turbulent flows in dredged trenches. *Journal of Hydraulic Research* 1995; **33**(6):813–824.

