Verification of Computer Flow Simulation in Confuser and Diffuser Channels

Ivan Komarov, Sergey Osipov, Andrey Vegera, Darya Kharlamova, and Aleksey Zonov

1 Introduction

Nowadays, many methods for computer simulation of viscous liquor flows are known. The main difference between the methods is the approach to solution of the Navier-Stocks equations that are 4 equations with 4 unknown variables [1, 2] at given boundary conditions. This system of differential equations together with the boundary and initial conditions is principally nonlinear, so it is impossible to build its analytical solution. The only available method is the Direct Numerical Simulation (DNS). This method consists of numerical solution of the nonstationary Navier–Stocks equations by application of space grids and time steps sufficient for the presentation of the flow structures. The DNS method requires grids with very small cells so it needs large computer resources. Nowadays, this factor limits its application by simple laminar flows at low values of Re < 103 [3–5]. In visible prospects, introduction of the DNS method into applied technical problems does not seem realistic.

In practice, the applied technical problems are solved with the widely spread method known as Reynolds-averaged Navier–Stokes, or RANS, and the problem closure with semi-empirical turbulence models. The known turbulence models are numerous. The most common are the two-parameter turbulence equations classified to low Reynolds $k - \omega$ and SST and high Reynolds $k - \varepsilon$ models group.

The turbulence model together with the grid parameters influences the correspondence of calculation results and physical experiments. The turbulence model mostly determines the dimensionless distance y^+ between the wall and the first calculation cell. The y^+ parameter recommended values are known for different turbulence models. The high Reynolds $k - \varepsilon$ models require the first cell to correspond with the y^+ in the range of $30 \leq y^+ \leq 100$ [6, 7]. In the low Reynolds SST/$k - \omega$ models, the first cell distance to the wall must be within $y^+ \leq 5$ [8].

I. Komarov · S. Osipov (✉) · A. Vegera · D. Kharlamova · A. Zonov
National Research University "Moscow Power Engineering Institute", 111250 Moscow, Russia

A. Irina and P. Zunino (eds.), *Proceedings of the International Symposium on Sustainable Energy and Power Engineering 2021*, Lecture Notes in Mechanical Engineering,
https://doi.org/10.1007/978-981-16-9376-2_33

Table 1 Review of papers devoted to the verification

Investigation object	y^+	Turbulence model	Paper
Pump	40	$k - \varepsilon$	[9]
Scrolled tube	1	SST	[10]
Radial turbine scroll channel	1	SST	[11]
Compressor duct	0.18–0.25	SST	[12]
Flat diffuser	1–15	SST	[13]
Combustor	1–5	RNG $k - \varepsilon$	[14]

The numerical experiment parameters that provide its minimal error are determined by experimental verification. Many published papers compare results of computer and physical experiments, but these papers do not always describe the complete set of the experiment parameters. Usually, attention is paid to the turbulence model and the y^+ parameter. Table 1 summarizes the results of the research literature review, turbulence models, and the y^+ parameter values that provide minimal computer experiment errors.

Usually, the tools for computer simulation of physical processes involve complicated models that suit the design of actual technical objects. At these conditions, the simulation result may be applied only to specific areas of the investigated object, but a transition to another design area may increase the simulation error which is caused by the large number of factors influencing the physical process. The computer tools application limits may be found in the simplified flow path of the technical object. The equipment flow path is decomposed into typical channels and the application limits of turbulence models and grid parameters are found in these simplified problems and flow parameters. The equipment elements simplification is not aimed at simulation of power industry problems by primary cases. The goal is to determine a similarity degree for problems and further verify the general approach to computer simulation. The simulation results' verification with the physical experiments in the simplified elements allows the development of general recommendations for simulation of similar problems, flow analysis in various diffuser and confuser channels, sudden expansion or throttling, etc. After the limits of computer experiment, are determined, that in complex channels the verification objects may be improved by combination of typical channels and the recommendations accuracy should be checked by the physical process simulation.

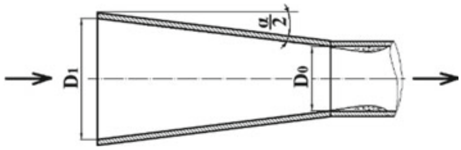

Fig. 1 Confuser scheme and its main geometry parameters, D_1, D_0—inlet and exit diameters, α—constriction angle

Table 2 Confuser geometry parameters for verification

No.	Geometry parameters			Flow parameters		
	α (°)	n_0	D_0 (mm)	Re		
1	3	0.64	50	100,000	200,000	400,000
2	3	0.39	50			
3	10	0.39	50			

2 Verification Results of the Computer Simulation for Flow Analysis in Confusor Channels

2.1 Investigation Object

Confuser or converging channels are widely used in various devices, centrifugal and axial compressors, jet pumps, cooling towers, fans, flow meters, etc. The confuser function is to produce a uniform flow velocity distribution in the equipment flow path. The confuser resistance coefficient at turbulent flow depends upon design and condition parameters. The geometry parameters are the constriction angle α (Fig. 1) and the constriction degree $n_0 = F_0/F_1$. The flow condition parameter is the Reynolds number Re.

An important stage of the recommendations issue for the flow analysis parameters is the selection of adequate physical experiment results with detailed descriptions of the test boundary conditions and the results analysis method. The computer experiment parameters with minimal simulation error were verified with the physical test results of the book [15]. The geometry and flow parameters taken from this book are summarized in Table 2.

2.2 Solver and Grid Parameters

The channel flow analysis involved the RANS averaging method and the turbulence models SST, $k - \omega$, and $k - \varepsilon$ Standard. The main grid and solver tuning verified parameters are the turbulence model, the y^+ parameter, the number of prismatic layers,

Table 3 Grid parameters

Parameter	Value
Grid type	Not structured
Global cell size (mm)	5
Prismatic grid growth law	Exponential
y^+ parameter	0.25 … 90
Prizmatic layers height ratio	1 (k − epsilon), 1.3 (k − omega, SST)
Number of layers	1 (k − epsilon), 10–15 (k − omega, SST)

Table 4 Flow simulation boundary conditions

Confuzer No.	G_0, G/s		
	Re = 100,000	Re = 200,000	Re = 400,000
1	89.08	178.279	114.101
2	113.920	229.166	228.202
3	114.101	458.333	456.405

and the growth coefficient. The fixed parameters are the global cell size, the prismatic grid growth, and the wall function. The wall area cell parameters depend upon the preliminary assumed turbulence model. The verified and fixed grid parameters and the solver tuning are summarized in Table 3.

In the physical experiment [2], the working fluid was air as an ideal gas. The channel inlet boundary conditions are the working fluid mass flow $G0$ determined in relation with the experiment value of Re criteria and $D0$ (Table 4). The inlet flow temperature is $T = 20$ °C and the exit static pressure is $P = 1$ bar.

2.3 *Verification of Flow Computer Simulation in Confuser Channels*

The computer simulation results are compared with the physical experiment taken for reference by the energy loss coefficient ξ. Figure 2 shows the confuser #1 ($\alpha = 3°$, $n_0 = 0.64$) simulation errors in relation to y^+ for the $k - \varepsilon$ turbulence model.

All dependencies above are similar, the larger values of y^+ correspond to larger errors of the flow computer simulation. The area of acceptable errors of 10% for the whole range of Reynolds numbers contains the simulation results at $y^+ = 10 - 30$.

Figure 3 presents relations of the diffuser #1 computer simulation errors with the y^+ values at the same range of Re and the SST turbulence model.

The relations between gradual error and y^+ are logarithmic with the gradual error growth at y^+ increase and the rapid decrease into the large negative values at y^+ approach

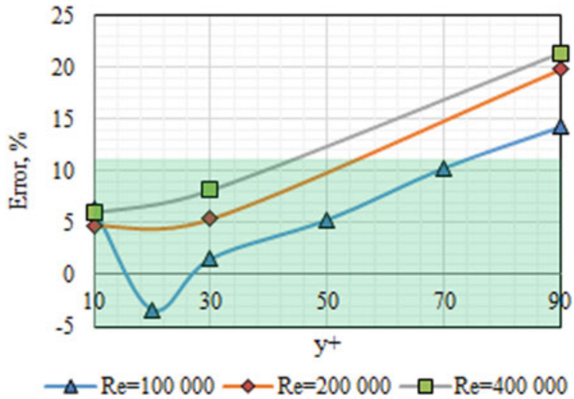

Fig. 2 Confuser No. 1 with $\alpha = 3°$, $n_0 = 0.64$, simulation error at $k -$ epsilon turbulence model versus the y^+ parameter

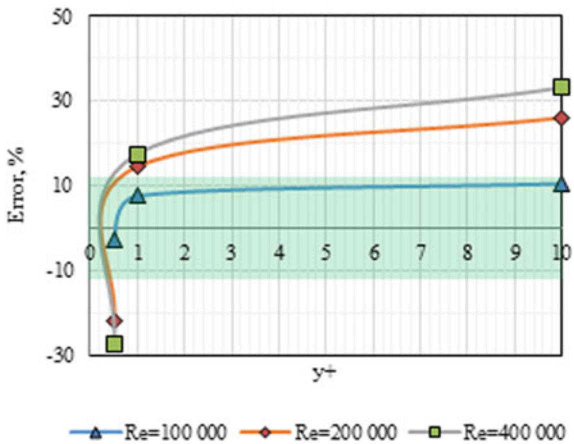

Fig. 3 Confuser No. 1 with $\alpha = 3°$, $n_0 = 0.64$, simulation error at SST turbulence model versus the y^+ parameter

to zero. Also, at large values of Re, the error grows faster with the y^+ increase. Comparison of the results of simulation with SST turbulence model with the reference data shows that the simulation error stays acceptable only at Re = 100,000 and $y^+ = 0.5$ and $y^+ = 1$. Thus, the SST turbulence model may be applied only in a limited range of the channel flow regimes.

Figure 4 presents dependencies of the confuser #1 flow simulation errors from the y^+ parameter in the same range of Re values at the $k - \omega$ turbulence model. The dependencies logarithmic configuration is similar to the described above one obtained with the SST turbulence model. The area of acceptable error of 10% contains the $y^+ = 1$ for the whole range of the values Re = 100,000–400,000.

The number of prismatic layers is an important parameter for the flow computer simulation with low Reynolds models. The influence of this parameter upon the analysis error is evaluated by flow simulation in confuser #1 at Re = 100,000, $k - \omega$ turbulence model, $y^+ = 1$, and verified number prismatic layers (Fig. 5).

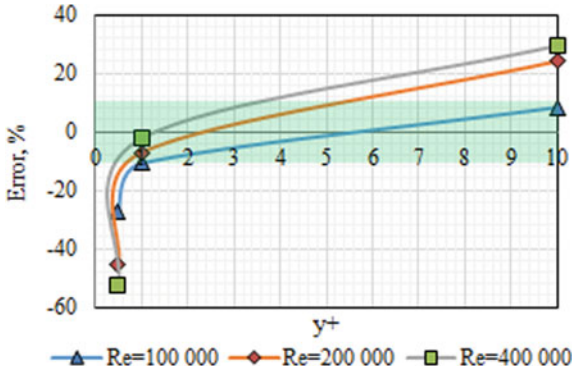

Fig. 4 Confuser No. 1 with $\alpha = 3°$, $n_0 = 0.64$, simulation error at $k - \omega$ turbulence model versus the y^+ parameter

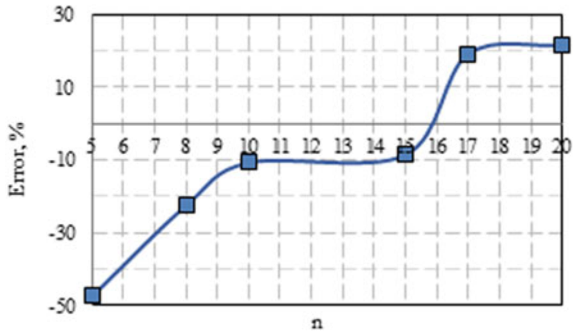

Fig. 5 Confuser No. 1 with $\alpha = 3°$, $n_0 = 0.64$, simulation error at SST turbulence model, $y^+ = 1$ versus the number of prismatic layers n

The simulation error dependence upon the number of prismatic layers has four specific bands. In the first band, the layers number increase from 5 to 10 is followed by the error reduction from 50 to 10%. In the second band, the layers number increase from 10 to 15 is followed by the absence of error sensitivity upon the number of layers. The further increase of the layers number from 15 to 17 increased the error from -10 to 19% and the further number increase up to 20 does not show any changes of error. So the optimal prismatic layers number in terms of the acceptable analysis error below 10% and minimal time needed for the grid buildup is 10.

Based on the analysis results for confuser #1 with $\alpha = 3°$, $n_0 = 0.64$, $D_0 = 50$ mm the analysis of confuser #2 with $\alpha = 3°$, $n_0 = 0.39$, $D_0 = 50$ mm was carried out with $k - \varepsilon$ and $k - \omega$ turbulence models. The verification shows that the $k - \varepsilon$ model gives the error above 20% at, $y^+ = 10 \ldots 30$. For the confuser #2, the minimal error in the range of Re = 10,000–400,000 is obtained at $y^+ = 0.75$. On the other side the confuser #3 simulation with $k - \omega$ turbulence model shows, that the necessary error was obtained with the y^+ parameter reduced down to 0.25. Thus, the $k - \omega$ model shows the widest application range for the confuser channel flow analysis. These results make the base for the further described verification of flow analysis in diffuser channels.

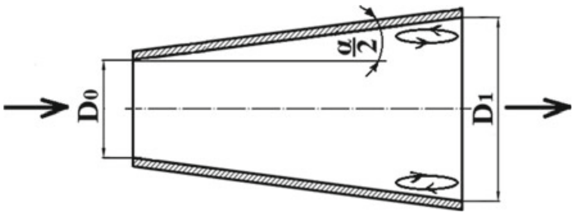

Fig. 6 Diffuser main dimensions, D_0, D_1—inlet and exit diameters, α—expansion angle

Table 5 Diffuser dimensions and flow parameters

No.	Dimensions			Flow parameters		
	α (°)	n_0	D_0, (mm)	Re		
1	14	4	80	100,000	200,000	400,000
2	29	6	80			

3 Verification of Flow Computer Simulation in Diffuser Channels

3.1 Investigation Object

A diffuser is a smoothly divergent channel or tube that provides a transition from a smaller cross-section station to a larger one (Fig. 6). Diffuser flow path elements are quite usual, so the problem of computer simulation in terms of minimal calculation error is of importance.

Diffuser dimensions and flow parameters for verification are taken equal to experimental models for the hydraulic resistance tests [12]. Table 5 summarizes dimensions and flow parameters taken for the analysis results verification, the flow has separation on the channel inner wall.

3.2 Solver and Grid Parameters

The diffuser flow simulation involves the method of Raynolds averaging Navier–Stocks (RANS) like the confuser flow. The turbulence simulation model is the $k-$ omega that shows a wider application range in confusers. The model grid parameters are presented in Table 6.

The physical experiment [2] is carried out with air as the ideal gas working fluid. The channel inlet boundary conditions working fluid G_0 mass flow and temperature $T = 20$ °C depend upon the experiment Reynolds number Re value and diameter D_0 (Table 7), the channel exit static pressure $P = 1$ bar.

Table 6 Diffuser model grid parameters

Parameter	Value
Grid type	Not structured
Glebal cell dimension, mm	8
Prizmatic grid growth law	Exponential
y^+ parameter	0.005 … 0.5
Prismatic layers ratio	1.3
Number of layers	10

Table 7 Flow simulation boundary conditions

Diffuser No.	G_0, G/s		
	Re = 100,000	Re = 200,000	Re = 400,000
1	113.965	227.930	455.860
2	113.965	227.930	455.860

3.3 Verification of the Flow Computer Simulation Results

Like in confusers (see above) the flow computer simulation results are compared against the reference ones by the hydraulic resistance coefficient ζ. Figure 7 presents dependencies of the resistance calculation errors upon the y^+ parameter at different Re values and $k - \omega$ turbulence model for the diffuser #1 with $\alpha = 14°$ and $n0 = 4$. Verification results for diffuser #2 with $\alpha = 20°$ and $n0 = 6$ are presented in Fig. 8.

The dependencies show the same tendency for the simulation error increase following the y^+ growth. On the other side, the y^+ values that provide acceptable error are 25–200 times smaller than the similar condition values in confusers. In the diffuser #1, with $\alpha = 14°$ and $n_0 = 4$ at Re = 100,000–400,000, the simulation error below 10% is possible at $y^+ = 0.03$–0.04. At the increase of the extension angle up to 20° and the expansion rate up to 6 in diffuser #2, the same simulation error of 10% is possible at $y^+ = 0.005$–0.01.

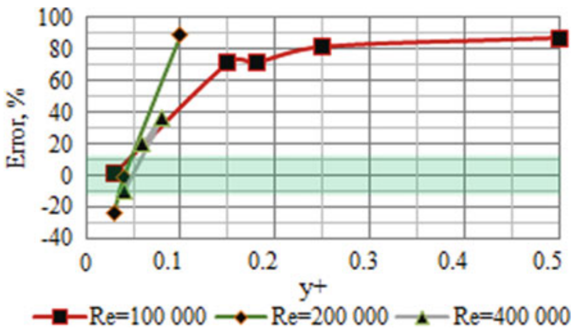

Fig. 7 Diffuser No. 1 simulation error at $k - \omega$ turbulence model versus the y^+ parameter

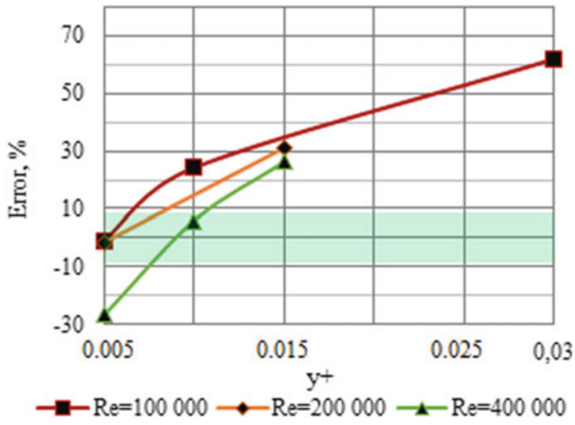

Fig. 8 Diffuser No. 2 simulation error at $k-\omega$ turbulence model versus the y^+ parameter

4 Conclusions

In flow computer simulation of confuser and diffuser channels the dimensionless parameter y^+ influences on the simulation error together with different turbulence models. The increasing of the y^+ parameter causes increasing of the calculation results errors at SST/ $k-\omega$ and $k-\varepsilon$ Standard turbulence models. The optimal number of prismatic layers for low Reynolds k − ω turbulence model with the simulation error below 10% and minimal model grid time spending is 10. In two ranges of layers number between 10 and 15 and between 17 and 20, the layers number does not influences the error. The $k-\omega$ turbulence model is applicable to the confuser and diffuser flow simulation with the simulation error below 10%. To keep the error below 10% at Reynolds number values Re = 100,000–40,000 in confuser channels, it is recommended to use the value $y^+ = 1$. In the flow simulation of diffusers with expansion angle 14–20° at Re = 100,000–40,000 the error of 10% may be obtained at $y^+ = 0.005$–0.04 and the larger is the expansion angle the smaller y^+ values must be used.

This study conducted by Moscow Power Engineering Institute was financially supported by the Ministry of Science and Higher Education of the Russian Federation (project No. FSWF-2020-0020).

References

1. Pozrikidis, C.: Fluid Dynamics: Theory, Computation, and Numerical Simulation. Springer, New York (2016)
2. Versteeg, H.K., Weeratunge, M.: An Introduction to Computational Fluid Dynamics: The Finite Method. Pearson education, London (2007)
3. Elghobashi, S.: Annu. Rev. Fluid Mech. **51**, 217 (2019)
4. Zhang, C., Duan, L., Choudhari, M.M.: AIAA J. **56**, 4297 (2018)
5. Anantharamu, S., Mahesh, K.: J. Fluid Mech. **898** (2020)

6. Ahsan, M.: Beni-Seuf Univ. J. Appl. Sci. **4**, 269 (2014)
7. Salim, S.M., Cheah, S.: Proceedings of the international multiconference of engineers and computer scientists, vol. 2, p. 2165 (2009)
8. Catalano, P., Amato, M.: Aerosp. Sci. Technol. **7**, 493 (2003)
9. Yang, S.S., Derakhshan, S., Kong, F.Y.: Renew. Energy **48**, 507 (2012)
10. Tang, X., Dai, X., Zhu, D.: Int. J. Heat Mass Transf. **90**, 523 (2015)
11. Hamel, M., Hamidou, M.K., Cherif, H.T., Abidat, M., Litim, S.A.: In: Proceedings of Turbo Expo: Power for Land, Sea, and Air, vol. 43161, p. 2329 (2008)
12. Gileva, L.V., Aksenov, A.A., Kozhukhov, Y.V., Petrov, A.Y.: In: AIP Conference Proceedings, vol. 1, p. 030038 (2020)
13. DalBello, T., Dippold III, V., Georgiadis, N.J.: NASA TM 2005-213894 (2005)
14. Patil, S., Abraham, S., Tafti, D., Ekkad, S., Kim, Y., Dutta, P., Srinivasan, R.: J. Turbomach 1 (2011)
15. Idelchik, I.E.: Handbook of Hydraulic Resistance. Washington (1986)