Criteria of computational grid generation for turbulence models taking into account laminar-turbulent transition

A A Matyushenko¹, A S Stabnikov¹ and A V Garbaruk¹
¹ Peter the Great St.Petersburg Polytechnic University, Politechnicheskaya 29, Saint-Petersburg 195251, Russia

E-mail: alexey.matyushenko@gmail.com

Abstract. Criteria of structured and unstructured mesh generation for turbulence models taking into account laminar-turbulent transition were developed using flat plate boundary layer flow. It is shown the first near wall cell should be placed in the viscous sublayer in the transition region and stretching factor should not exceed 1.4. For the unstructured meshes all the boundary layer should be covered by prismatic layers. The results of the prediction of the flow around a NLF(1)-0414F airfoil show that the use of relatively coarse meshes provide acceptable accuracy of the solution.

1. Introduction
In the numerical study of flows where the laminar-turbulent transition plays a significant role, the Reynolds equations should be solved in combination with the turbulence models which can predict laminar-turbulent transition (so-called transition models). These type of models are relatively «young», so the recommendations for building computational grids, which are available in the literature, are very imperfect. This determined the goal of the present study, which consists in development of the criteria for generation of the structured and unstructured computational grids providing acceptable accuracy of solution for transition models.

2. Considered transition models and CFD codes
Numerical simulations were carried out solving incompressible Reynolds-averaged Navier-Stokes (RANS) equations in combination with three transition models solving additional equations together with $k-\omega$ SST [1] or SA [2] turbulence models. The considered transition models are based on solving additional transport equation for the turbulence intermittency ($\gamma$-$Re_{\theta}$ SST [3], $\gamma$-SST [4] and $\gamma$-SA [5]) for the prediction of transition location.

Commercial ANSYS Fluent and the academic NTS (Numerical Turbulence Simulation) CFD codes with different types of grids were used for the study. For ANSYS Fluent structured and hybrid unstructured grids were generated using ICEM CFD and ANSYS Meshing correspondingly whereas block «chimera-type» structured grids were used in the NTS code. SST based transition models ($\gamma$-$Re_{\theta}$ SST and $\gamma$-SST) are implemented in both considered CFD codes, while SA based model ($\gamma$-SA) is available only in the NTS code.

The double precision version of the solvers was used in all the computations. For all transport equations, including the equations for turbulent variables, order of the discretization scheme for the convective terms was not less than two. For each test case, all simulations were started from the same uniform initial
guess and the simulations were performed until the normalized maximum residuals of all the equations were below a value of $10^{-5}$.

3. Criteria development for transitional boundary layer

The study was carried out on the structured and unstructured grids for zero pressure gradient transitional flat plate boundary layer using ANSYS Fluent. The Reynolds number based on incoming uniform flow and flat plate length was $Re=3\cdot10^6$ [6]. The $\gamma$-$Re_\theta$ SST and $\gamma$-SST transition models were considered. The rectangle computational domain shown in Figure 1 was 2 [m] length and 0.5 [m] height. A constant velocity was specified at the inlet section of the computational domain. No-slip conditions were used on the wall and constant pressure was specified on the outlet. Symmetry boundary condition was specified on the upper boundary. Inlet turbulence characteristics ($Tu=3\%$ and $v_t/\nu=12$) were specified in order to reproduce experimental freestream turbulence kinetic energy along the flat plate.

![Figure 1](image1.png)

**Figure 1.** Computational domain with boundary conditions.

1.1. Structured grids

For this type of grids three parameters were varied: streamwise resolution in the transitional area, the height of the first near wall cell and geometric progression expansion ratio (ER) of the grid in normal-wall direction. Figure 2 shows that mesh with streamwise grid step $\Delta X = 10\delta$ ($\delta$ - boundary layer thickness) provides fine solution. Thus the usual streamwise grid step requirements for fully turbulent models are acceptable for the transition models.

In case of first near wall height variation the transition location is insensitive to $\Delta Y_1^+<1$ in the transition region (Figure 3). For higher values of $\Delta Y_1^+>1$ in this area transition location moves upstream. On the other hand sensitivity of the transition location to the expansion ratio is not so strong and slightly depends on the choice of the transition model (Figure 4).

![Figure 2](image2.png)

**Figure 2.** Effect of the resolution of transition region in streamwise direction for the considered turbulence models.
Based on the above study the following recommendation for mesh generation are considered. The first near wall grid step should be placed into the viscous sublayer of the boundary layer in the transition region \((Y^+ ≤ 1)\), and the grid expansion ratio in normal to the surface should not exceed 1.4. The grid step in streamwise direction can be ten times higher than boundary layer thickness.

### 1.2. Unstructured grids

The hybrid unstructured grids are increasingly used in engineering practice. For such meshes near the wall surface prismatic layers are grown in normal wall direction and the rest of the domain fill with tetrahedral/polyhedral cells. For the considered testcase several grids were generated with different number of prismatic layers fully or partially covering boundary layer along the flat plate. For the coarsest mesh prismatic layers cover only 12% of the boundary layer at the outlet boundary while fine meshes fully cover it. An example of the mesh structure in the transition region is shown in Figure 5.
Figure 5. Mesh structure in the transition region for unstructured mesh.

Figure 6 shows the comparison of the skin friction coefficient for the considered meshes. Meshes almost and fully covered the boundary layer \((PL=0.87\text{ and } PL=2.27)\) provide grid-independent solution. Medium mesh \((PL=0.33)\) predicts only laminar and transitional part of the flow. In the area where prismatic layers partially cover the boundary layer \((X>1.2 \text{ [m]})\) the mesh provides wrong solution. Thus in the case of unstructured meshes prismatic layers should completely cover the boundary layer.

Due to poor resolution of the boundary layer along entire flat plate solution was not converged on the mesh with \(PL=0.12\).

| \(\gamma\)-Re\(_{\text{eff}}\) SST | \(\gamma\)-SST |
|----------------|----------------|
| ![Graph](image1.png) | ![Graph](image2.png) |

Figure 6. Comparison of the skin friction coefficient for different transition models on the considered unstructured meshes. \(PL=H_{PL}/\delta\) \((H_{PL} \text{ and } \delta \text{ is the prismatic layer height and boundary layer thickness at } X=2 \text{ [m]})\).

4. **Prediction of the flow around a NLF(1)-0414F airfoil in wide range of angle of attack**

The considered criteria were applied for prediction of flow around a NLF(1)-0414F airfoil [7] at Reynolds number \(Re = U_{\infty} \cdot C/\nu = 3 \cdot 10^6\) based on incoming uniform flow and airfoil chord. For this flow regime laminar-drag bucket phenomenon (the minimum drag coefficient extends over a range of lift coefficient) is observed for low angles of attack \((\alpha = -5^\circ \text{ to } 8^\circ)\). Prediction of such phenomenon strongly depends on the prediction of the transition location on the airfoil.

Numerical simulations were carried in the two dimensional wind tunnel which height \(H=2.5C\) corresponds to the experimental size (Figure 7). The angle of attack was specified by rotation of the airfoil in the tunnel. Inlet and outlet boundaries were located 10\(C\) upstream and downstream of the leading edge of the airfoil. A constant velocity was specified at the inlet section of the computational domain. No-slip conditions were used on the airfoil boundary and constant pressure was specified on the outlet. Symmetry boundary conditions were specified on the wind tunnel upper and lower walls for imitation of the slip-walls. For \(k-\omega\) based models the inlet turbulent kinetic energy provided
experimental low turbulence intensity $T_u = 0.08\%$ near the leading edge of the upstream airfoil and the specific dissipation rate was specified as $\omega = 10 \cdot U/\sigma = 10[1/s]$. For the SA-based model only the analogous value of turbulent viscosity was specified. Four computational grids with different resolution of the boundary layer in normal-wall direction were used (Table 1). The fine structured mesh with low expansion ratio $ER = 1.1$ ensures $\Delta Y_{+1,max} < 1$. The other meshes were generated with above described recommendations: $\Delta Y_{+1}$ was less than 1 only in transition region and $ER = 1.4$. In case of unstructured mesh twenty prismatic layers covering boundary layer along the airfoil were grown. Figure 7 shows an example of the structured and unstructured mesh in the vicinity of the airfoil.

### Table 1. Mesh parameters for flow of flow around the NLF(1)-0414F airfoil.

| Mesh   | CFD Code     | Type       | $\Delta Y_{+1}$ in the transition region | ER  |
|--------|--------------|------------|----------------------------------------|-----|
| Mesh-1 | ANSYS Fluent | Structured | 0.2                                    | 1.1 |
| Mesh-2 | ANSYS Fluent | Structured | 1                                      | 1.4 |
| Mesh-3 | ANSYS Fluent | Unstructured | 1                                      | 1.4 |
| Mesh-4 | NTS Code     | Structured | 1                                      | 1.4 |

Figure 7. Computational domain, boundary conditions and mesh structure for prediction of the flow around the NLF(1)-0414F airfoil.
Figure 8 shows that all the models predict laminar drag bucket for all the computational meshes. Globally the predicted airfoil characteristics are in good agreement with each other for all turbulence models and both CFD codes.

![Figure 8](image-url)

Figure 8. Comparison of experimental and computational sectional lift coefficient on the considered meshes in a wide range of angles of attack.

5. Conclusions
Criteria of structured and unstructured mesh generation for different turbulence models taking into account laminar-turbulent transition were developed. For structured grids the first near wall grid step should fall into the viscous sublayer of the boundary layer in the transition region ($Y^+ \leq 1$), and the grid expansion ratio in normal to the surface should not exceed 1.4. Finally, in the case of unstructured meshes, prismatic layers should completely cover the boundary layer.

Acknowledgements
The reported study was funded by RFBR, project number 19-31-90046.
The results of the present work were obtained using computational resources of Peter the Great Saint-Petersburg Polytechnic University Supercomputing Center (http://www.spbstu.ru).

References
[1] Menter F R, Kuntz M and Langtry R B 2003 Ten years of experience with the SST turbulence model *Turbulence, Heat and Mass Transfer* 4 625–32
[2] Spalart P R and Allmaras S R 1994 A one-equation turbulence model for aerodynamic flows *La Recherche Aerospatiale* 1 5-21
[3] Langtry R B and Menter F R 2009 Correlation-based transition modeling for unstructured parallelized computational fluid dynamics codes *AIAA Journal* 47 (12) 2894–906
[4] Menter F R, Smirnov P E, Liu T and Avancha R 2015 A one-equation local correlation-based transition model *Flow Turbulence Combust* 95 583–619
[5] Coder J M and Maughmer M D 2012 One-equation transition closure for eddy-viscosity turbulence models in CFD *AIAA Journal* 52 (11) 2506-12
[6] Savill A M 1993 Some recent progress in the turbulence modelling of by-pass transition *Near-Wall Turbulent Flows* ed C G Speziale and B E Launder (New York: Elsevier) pp 829-48
[7] McGhee R, Viken J, Pfenninger W, Beasley W and Harvey W 1984 Experimental results for a flapped natural-laminar-flow airfoil with high lift/drag ratio *NASA Technical Memorandum* 85788