Unsteady Flow Over a Smooth Flat Plate Using DES
S. Mozaffari, Michel Visonneau, J. Wackers

To cite this version:
S. Mozaffari, Michel Visonneau, J. Wackers. Unsteady Flow Over a Smooth Flat Plate Using DES. 20th Numerical Towing Tank Symposium (NuTTS 2017), Oct 2017, Wageningen, Netherlands. hal-02569766

HAL Id: hal-02569766
https://hal.science/hal-02569766
Submitted on 11 May 2020

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L’archive ouverte pluridisciplinaire HAL, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d’enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.
1 Introduction

Computations based on Reynolds-Averaged Navier–Stokes (RANS) equations are common in industry today. Although they are very successful in predicting many parts of the flow around a vehicle, a ship or an airplane, they still have some limitations for the high Reynolds number flows. Detached Eddy Simulation (DES) is a modification of a RANS model for the prediction of turbulent flows at high Reynolds numbers. It was first proposed as a solution to the high computational costs of applying Large-Eddy Simulation (LES) to complete configurations at high Reynolds numbers [Spalart et al., 1997], and to improve the predictive capabilities of turbulence models in highly separated regions [Guilmineau et al., 2011].

On the other hand, adaptive mesh refinement (AMR), is a solution to accurately capture flow features. It is a technique to speed up the computation by starting the solution on a coarse grid and refining this grid locally to accurately resolve areas of interest. It is an ideal way to efficiently solve flow problems that have strong local structures whose position is not known a priori without excessive increase in computational effort.

Application of both methods is the subject of many recent studies. One of the issues confronting using AMR method is the creation or destruction of the turbulence at the interface of the coarse and fine grids, where the turbulent properties’ conservations are violated. Using the DES model, would make the case more complex, especially at the vicinity of RANSE/LES interface because of the existence of a grey area in which a shear layer, after separation, must generate LES content (random eddies) which it did not possess in the boundary layer upstream [Squires, 2004]. Thus, in our project, first of all, in order to look at the grey zone and its effect, the DES method for a turbulent flow over a smooth flat plate is investigated.

In the section 2, the DES used in the ISIS-CFD flow solver as a part of the FINE™/Marine computing suite, is formulated in detail. Then the physical description, computational domain, grid generation and the numerical setup of the case are explained. In the computation part, RANSE and DES turbulence models are compared, in order to see which one gives a better representation of the physical phenomena.

2 Detached Eddy Simulation (DES)

The DES version used in ISIS-CFD, is based on the $k$-$\omega$ SST model, because it is consistently considered as one of the best two-equation RANS models, particularly for separation prediction. The DES modification is applied to the dissipation term in the $k$ transport equation. In the original model, the dissipation term is written as:

$$\rho \varepsilon = \beta^* \rho K \omega$$  

where $\varepsilon$ is the dissipation rate, $\beta^*$ is a constant of the SST model. For the SST-DES, this term is written now as:

$$\rho \varepsilon = \beta^* \rho K \omega F_{DES}$$  

with

$$F_{DES} = \max\left(\frac{L_t}{C_{DES} \Delta}, 1\right)$$

where $\Delta$ is the maximum local grid spacing ($\Delta = \max(\Delta x, \Delta y, \Delta z)$), $L_t$ is the turbulent length scale, $L_t = \sqrt{K/(\beta^* \omega)}$ and $C_{DES}$ is a constant. In the initial version of the DES-SST [Menter et al., 2003], $C_{DES}$ was 0.61. For the ISIS-CFD solver, we prefer to use the value $C_{DES} = 0.78$ [Menter and Kuntz., 2003]. According to the above equation, wherever $F_{DES} > 1$, LES is used. In contrast for $F_{DES} \approx 1$, all turbulence is modeled and RANSE is used.
3 Case Setup

The computational domain sizes in the streamwise, wall normal and spanwise directions are respectively $L_x = 0.5m$, $L_y = 0.1m$ and $L_z = 0.005m$. Three meshes were generated according to the data in Table 1. The meshes were refined only in the streamwise direction. Moreover, another domain with a larger size in spanwise direction ($L_x = 0.5m$, $L_y = 0.1m$ and $L_z = 0.05m$) is also taken into consideration to study the possible turbulent effects over the flow in spanwise direction. The parameters of this mesh (called Mesh 4) are tabulated and presented in Table 1. The first layer thickness in wall normal direction is $1.59167 \times 10^{-6}m$ and constant for all the meshes. The free stream velocity is $U_e = 70ms^{-1}$, and the Reynolds number per meter is $Re = 4.72 \times 10^6m^{-1}$. The computation is unsteady and the time step for all the computations was set to $10^{-4}s$.

4 Results & Discussions

In this part, the results of the computations which have been done according to the previous explanations, are presented. To get a broader view of the grids in the boundary layer, Fig. 1 shows the grid densities scaled using the boundary layer thickness ($\delta$) at $X = 0.5m$ for each mesh.

<table>
<thead>
<tr>
<th>Mesh name</th>
<th>$N_x \times N_y \times N_z$</th>
<th>$y^+$</th>
<th>Streching ratio</th>
<th>$N_{cells}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh 1</td>
<td>$300 \times 137 \times 50$</td>
<td>0.2</td>
<td>1.05</td>
<td>2 055 000</td>
</tr>
<tr>
<td>Mesh 2</td>
<td>$800 \times 137 \times 50$</td>
<td>0.2</td>
<td>1.05</td>
<td>5 480 000</td>
</tr>
<tr>
<td>Mesh 3</td>
<td>$1500 \times 79 \times 50$</td>
<td>0.2</td>
<td>1.1</td>
<td>5 925 000</td>
</tr>
<tr>
<td>Mesh 4</td>
<td>$1000 \times 94 \times 100$</td>
<td>0.2</td>
<td>1.08</td>
<td>9 400 000</td>
</tr>
</tbody>
</table>

Reynolds number per meter is $Re = 4.72 \times 10^6m^{-1}$. The computation is unsteady and the time step for all the computations was set to $10^{-4}s$.

It is seen in Table 2 that by refining the grid in streamwise direction, the boundary layer thickness is decreasing, and for Mesh 4, it is almost half of the value computed by the RANSE model. In addition, Fig. 2 shows the instantaneous velocity profile in the streamwise direction. Apart from the boundary layer thickness changes, no unsteadiness and fluctuation was captured.

According to Fig. 3, on finer meshes, the $F_{DES}$ function increases as it should. According to section 2, this switches the turbulence model to LES, so we would expect that the resolved part of the solution...
Table 2: Boundary layer thickness evolution (at $X = 0.5m$)

<table>
<thead>
<tr>
<th>Mesh name</th>
<th>$\delta$(m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh 1</td>
<td>5.13 E-03</td>
</tr>
<tr>
<td>Mesh 2</td>
<td>3.47 E-03</td>
</tr>
<tr>
<td>Mesh 3</td>
<td>2.97 E-03</td>
</tr>
<tr>
<td>Mesh 4</td>
<td>3.40 E-03</td>
</tr>
<tr>
<td>Mesh 4 (RANSE)</td>
<td>6.31 E-03</td>
</tr>
</tbody>
</table>

Reviewing the presented results, several obvious deductions can be made:

- No unsteadiness is seen in the results.
- Flow rate is negligible in Z direction (spanwise velocity is zero).
- The averaged resolved turbulent stresses are also zero.
- As the mesh is refined, the boundary layers thickness is reduced.
- For Mesh 4, the boundary layers thickness of RANSE computation is almost two times larger than the one in DES case.

5 Conclusion

A real flat-plate boundary layer is unsteady, but in this case there no unsteadiness was captured by the DES model. It will have impact on grid adaptation. One possibility which should be tested, is imposing unsteadiness into the system. In the next step of the study, using a stochastic approach, at each time-step, an unsteady velocity fluctuation is generated and will be used as the inlet boundary condition of the domain (Fig. 5), to see if it would be helpful to have resolved solution for the part of the part of the domain where the turbulence model is switched to LES.
Fig. 3: $F_{DES}$ Function
(a) modeled
(b) resolved

Fig. 4: Averaged Turbulent Stress $u'u'$ of the DES simulation for Mesh 4

References


Fig. 5: Inlet generated velocity at two different time-step