Hybrid LES/URANS simulation of turbulent natural convection by BEM

P. Kocutar, L. Škerget, J. Ravnik*

Faculty of Mechanical Engineering, University of Maribor, Smetanova 17, SI-2000 Maribor, Slovenia

A R T I C L E   I N F O

Article history:
Received 25 September 2014
Received in revised form 23 April 2015
Accepted 12 June 2015

Keywords:
Subdomain boundary element method
Hybrid LES-RANS
Turbulent flow
Heat transfer
Natural convection

A B S T R A C T

In this paper we have developed a hybrid LES/URANS turbulent model for a BEM based turbulent fluid flow solver. We employed the unified LES/URANS approach, where the interface between the LES and URANS regions is defined using a physical quantity, which dynamically changes during numerical simulation. The main characteristic of the unified hybrid model is that only one set of governing equations is used for fluid flow simulation in both the LES and URANS regions. Regions where turbulent kinetic energy is calculated by LES and URANS models are determined using a switching criterion. We used the Reynolds number based on turbulent kinetic energy and the Reynolds number based on total turbulent kinetic energy to establish the LES/URANS interface switching criterion. Depending on flow characteristics and with the use of switching criterion, we chose between sub-grid scale viscosity (SGS) and URANS effective viscosity. The SGS or URANS effective viscosity is used in the transport equation for turbulent kinetic energy and in governing equations for fluid flow. The developed numerical algorithm was tested by simulating turbulent natural convection within a square cavity. The hybrid turbulent model was implemented within a numerical algorithm based on the boundary element method, where single domain and sub-domain approaches are used. The governing equations are written in velocity-vorticity formulation. We used the false transient time scheme for the kinematics equation.

© 2015 Elsevier Ltd. All rights reserved.

1. Introduction

Turbulence occurs in fluid flows of high Reynolds numbers. It is present in the majority of fluid machines and devices, and is simulated during current engineering practise. Turbulence can be simulated by directly solving Navier–Stokes equations; this approach is called Direct Numerical Simulation, DNS. When using the DNS, we have to simulate flow phenomena that span the entire energy spectrum. As, at high Reynolds numbers, the Kolmogorov length scale is very small compared to the characteristic length scale of the flow, very dense meshes are required and thus high computational resources are demanded. As the DNS is very expensive from a computational point of view, it is currently being used mainly for research and to simulate benchmark cases. In the case of Large Eddy Simulation – LES – only those large eddies with the most energy are being simulated, while smaller eddies that cannot be described by the mesh are being modelled. Turbulent flow can also be simulated with Reynolds Averaged Navier Stokes equations using RANS and unsteady RANS (URANS) models. Despite some disadvantages, URANS models are more widely used in turbulent fluid flow simulations and have become state-of-the-art in the CFD and engineering industry. URANS models demand reasonable computational resources.

URANS is successful when modelling thin, fully turbulent, locally attached boundary layers. In the near wall regions due to small turbulent structures LES requires very fine meshes. In the bulk flow, where the Kolmogorov length scale is larger, fine meshes are adequate. RANS only provides statistical information of the flow, therefore its simulation results lack important information, e.g. coherent structures and dominant frequencies. LES directly solves large-scale motions, therefore it also provides significant amounts of accurate information. RANS requires reasonable computational times, while the computational costs of LES are too high [7].

As both LES and URANS have certain advantages and disadvantages hybrid models have been introduced which combine these two models. The basic concept of LES/URANS hybrid turbulent models is to use LES within the bulk region, where LES provides better results than URANS, and to use URANS in the near wall regions, where the mesh is not dense enough to capture the flow structures required by LES. Although the concept of combining LES and URANS turbulent models is not new [25], there are also several open topics, e.g. coupling technique between LES and URANS regions, which quantity to use for the switching criterion, and which LES and URANS turbulent model is the more appropriate.
LES/URANS hybrid turbulent models can be divided into two approaches when selecting LES and URANS regions, segregated\cite{16,4,8} and unified\cite{9}. In regard to the segregated LES/URANS approach, the computational domain is divided before simulation into LES and URANS regions\cite{28}. Using this approach, LES and URANS are solved separately.

Unified models are also known as global or universal models. The main characteristic of unified models is that they use the same set of governing equations for fluid flow simulation for both LES and URANS. Unified hybrid models are further divided according to a switching criterion between LES and URANS regions. This switching criterion can be defined as a hard interface, where the user defines it within geometry, or the interface is defined dynamically during the simulation according to the switching criterion. One of the first steps when combining LES and URANS turbulent models was turbulent flow simulation using Detached-Eddy Simulation DES\cite{25}. The idea is to model the whole boundary layer where the attached eddies occur with the URANS models, and only separate region where the eddies are detached, with the LES model. The same concept was also used by Paik et al.\cite{14}. For the DES model, both researchers used the Spalart–Almaras one-equation turbulent model for turbulent viscosity. Breuer et al.\cite{2} used a unified LES/URANS hybrid turbulent model for simulation of turbulent channel flow and flow over a periodic hill. Davidson\cite{3} used a partially averaged Navier–Stokes (PANS) model for a RANS/LES hybrid model. This hybrid model was tested in a channel flow and in a hump flow using embedded LES. Schmidt and Breuer\cite{22} used a unified LES/URANS hybrid model utilizing a switching criterion based on turbulent kinetic energy. Simulation was performed on the 3-D diffuser flow and flow passed an airfoil. Hybrid models were also used for heat transfer problems. Kenjereš and Hanjalić\cite{10} compared LES, T-RANS and the hybrid RANS/LES turbulent model for simulating thermal convection between two flat horizontal walls cooled from above and heated from below, and natural convection within a square cavity.

Our work is based on the use of BEM for fluid flow. Škerget and Rek\cite{27} used the boundary-domain integral method for numerical solutions for incompressible fluid flow within a driven cavity. They used the velocity–vorticity formulations of governing equations for a simulation and false transient approach for the kinematic equation. Ravnik et al.\cite{18} used LES for the simulation of natural convection within a rectangular enclosure. BEM has been used for the solution of the kinematic equation and is combined with FEM, which is employed for the kinetics equation. Ramšak and Škerget\cite{17} studied thermal flow simulation using the multidomain boundary element method with mixed boundary elements and a subdomain technique. For turbulent flow simulation they used a two-equation $k–\epsilon$ turbulence model. Lupšće et al.\cite{12} studied turbulent flow using a velocity–vorticity form of governing equation and BEM. For the turbulent channel flow and backward facing step they used Spalart–Allmaras, Chien and Abe–Kondoh–Nagano turbulent models.

We have developed a hybrid LES/URANS turbulent model using BEM using a combination of single-domain and sub-domain approaches. We have used the LES/URANS hybrid model for non-stationary non-isothermal turbulent flow simulation. Using unified hybrid model, LES and URANS models are unified within transport equation for turbulent kinetic energy $k$ and are being solved simultaneously. The LES and URANS regions are not defined before simulation but are dynamically defined during simulation itself. Therefore the user also avoids problems of unknown fluid flows. The numerical algorithm switches to URANS mode within the near wall region and to the LES mode in the bulk flow. Subgrid viscosity is used in LES mode, while in URANS mode, URANS modelled viscosity is used.

A 2D benchmark simulation of natural convection in square cavity was used to test our BEM implementation of the hybrid model. We choose the benchmark simulation of natural convection in square cavity, because it is geometrically very simple and the boundary conditions are well defined and at the same time exhibits fully turbulent flow. Thus, we could focus our attention to the implementation of the hybrid model, such as determining of the LES/URANS region and switching criterion. Furthermore, the closed cavity case makes it easier to implement the hybrid model, as the model depends on the wall distance, which is well defined in this case. In the future, we plan to extend this work to other turbulent flows.

2. Governing equations

2.1. Velocity–vorticity formulation

The governing equations were written in the velocity–vorticity formulation. The fluid is considered to be incompressible fluid with constant molecular viscosity $\nu_m$ and constant heat diffusivity $\alpha_m$. We employed the velocity–vorticity numerical scheme with a false transient time scheme for the kinematic equation.

Governing equations for kinematics and kinetics have to be filtered for LES simulation and averaged for URANS simulation. After averaging/filtering governing equations we introduce a turbulent viscosity. Within the URANS region it is calculated by the URANS model and in LES by the SGS model. In the unified hybrid turbulent model we solve only one set of governing equations. Due to simplicity, we use a further derivation of equation notation without filtered or averaged signs.

Filtered/time averaged governing equations for fluid flow can be defined with effective kinematic viscosity $\nu_{ef}$ and effective heat diffusivity $\alpha_{ef}$. Effective kinematic viscosity is the sum of molecular and turbulent kinematic viscosity $\nu_{ef} = \nu_m + \nu_{t}$, effective heat diffusivity is the sum of molecular and turbulent heat diffusivity $\alpha_{ef} = \alpha_m + \alpha_t$. The kinematics equation is the curl of the vorticity vector written as a pseudo-transient problem to allow an under relaxation approach, while the kinetics equation is the vorticity equation found by taking the curl of the momentum equation. The equation for kinematics for incompressible plane flow can be written in velocity–vorticity formulation as\cite{12}

\[
\frac{\partial \omega_{ij}}{\partial x_i} + \frac{1}{\alpha} \frac{\partial \omega_{ij}}{\partial t} + e_{ijk} \frac{\partial \omega_{kl}}{\partial x_k} = 0,
\]

where $\alpha$ is the under-relaxation parameter. The kinetics equation is written for the transport of vorticity, using the curl of the second extended form of the momentum conservation equation in primitive variables formulation\cite{29}. It can be written as\cite{21}

\[
\frac{D\omega}{Dt} = \nu_{ef} \frac{\partial^2 \omega}{\partial x_i \partial x_i} + e_{ef} (g \beta_g (T-T_0)) - e_{ef} \frac{\partial f_{ij}^m}{\partial x_i},
\]

where $T_0$ is a reference temperature, $g$ is a thermal volumetric expansion coefficient, $\beta_g$ is a gravity force oriented in opposite $y$ direction, $\omega$ is the $z$ component of the vorticity vector and $f_{ij}^m$ is a term defined as

\[
f_{ij}^m = -e_{ef} \frac{\partial \nu_{ef}(\omega)}{\partial x_i} + 2e_{ef} \frac{\partial \nu_{ef}}{\partial x_i} \frac{\partial \nu_{ef}}{\partial x_j} + 2e_{ef} \frac{\partial \nu_{ef}}{\partial x_j} \frac{\partial \nu_{ef}}{\partial x_i} + \nu_{ef} \frac{\partial \nu_{ef}}{\partial x_i} \frac{\partial \nu_{ef}}{\partial x_j}.
\]

Since we simulate an incompressible fluid, therefore the only force acting on fluid in natural convection flow is buoyancy. The buoyancy effect is considered in the kinetic equation\cite{2} by using Boussinesq’s approximation. For solving non-isothermal turbulent flow, we used a linear relationship of turbulent thermal diffusivity $\alpha_t$ and turbulent viscosity $\nu_t$, defined by the turbulent Prandtl number $Pr_t$ as $\alpha_t = \frac{\nu_t}{Pr_t}$. 
3. Hybrid LES/URANS turbulent model

3.1. Turbulent model for LES

The LES subgrid scale viscosity is calculated as Pope [15]. Filter width $\Delta$, dissipation part $\nu_{DGS}$ and turbulent viscosity $\nu_{SGS}$ for the sub-grid model can be written as

$$\nu_{SGS} = C_D \cdot k^{1/2} \cdot \Delta, \quad \nu_{DGS} = C_D \cdot k^{3/4} / \Delta, \quad \Delta = (\Delta x \cdot \Delta y)^{1/2},$$

where $C_D = 1.0$ and $C_D = 0.05$ [23].

3.2. Turbulent model for URANS

3.2.1. Rodi URANS model

We used the model developed by Rodi et al. [19]. This model is written using the following equations. Reynolds number defined with turbulent kinetic energy $k$ can be written as $Re_k = k^{1/2} \cdot y / \nu$, where $y$ is the normal distance from the wall. Turbulent viscosity can be written as $\nu_{URANS} = (\nu_2 / 3)^{1/2} \cdot l_{D,xy}$, characteristic length as $l_{D,xy} = C_{L,xy} \cdot y$, where $C_{L,xy} = 0.33$ and the normal velocity fluctuations $\sqrt{\nu_2}$ as $\nu_2 = k \cdot (4.65 \cdot 10^{-5} Re_k^2 + 4.00 \cdot 10^{-4} Re_k^2)$. Empirical corrections for the dissipation part, $D_{URANS}$, which correlates velocity scale, length scale and time scale, can be written as $D_{URANS} = (\sqrt{\nu_2})^{1/2} \cdot k / l_{D,xy}$, and dissipation length as $l_{D,xy} = 1.3 \cdot y / (1 + 2.12\nu / \sqrt{\nu_2})^{1/2}$. These equations are valid up to $Re_k \leq 60$.

3.2.2. High Reynolds number URANS model

Breuer et al. [2] used DNS simulation results from Moser et al. [13] for modification of the Rodi one-equation turbulent model presented in Rodi et al. [19]. It should be noted that this is not a high Reynolds turbulent model. The modified one-equation turbulent model can be written using the following equations. Normal wall velocity fluctuation $\sqrt{\nu_2}$ as $\sqrt{\nu_2} = k \cdot (3.55 \cdot 10^{-5} Re_k^2 + 6.50 \cdot 10^{-4} Re_k)$, where $C_{L,xy} = 0.4$ and dissipation length $l_{D,xy}$ as $l_{D,xy} = 1.5 \cdot y / (1 + 7.65\nu / \sqrt{\nu_2})^{1/2}$.

3.3. LES/URANS switching criterion

In this section we describe the switching criterion for defining LES and URANS regions. The definition is based on physical quantity.

3.3.1. Switching using Reynolds number defined by turbulent kinetic energy $k$

In the numerical algorithm we used the Reynolds number, for the switching criterion defined by the turbulent kinetic energy $k$, noted as $Re_k$, $Re_k$ can be defined by turbulent kinetic energy $k$, viscosity $\nu$, and the normal distance from the wall $y$ as

$$Re_k = k^{1/2} \cdot y / \nu.$$

In this way, switching criterion can be written as [2]

$$Re_k \leq C_{\text{switch}} \rightarrow \text{RANS mode},$$

$$Re_k > C_{\text{switch}} \rightarrow \text{LES mode},$$

where the value of $C_{\text{switch}}$ can be used up to 60.

3.3.2. Switching using Reynolds number defined by the total turbulent kinetic energy $k_{tot}$

The next switching criterion used in the hybrid model was the Reynolds number defined as total turbulent kinetic energy, $Re_{tot}$, which is defined similar to $Re_k$:

$$Re_{tot} = k_{tot} \cdot y / \nu,$$

where $k_{tot}$ is defined as the sum of the modelled turbulent kinetic energy, which is being solved in the transport equation (5), and the resolved turbulent kinetic energy, $k_{res}$, which is calculated from the simulated velocity field. The total turbulent kinetic energy $k_{tot}$ can be written as $k_{tot} = k + k_{res}$, where $k_{res}$ is defined as $k_{res} = (\nu_1')^2 / \rho$.

Part $\nu_1'$ in the equation for $k_{res}$ is the difference between the averaged and current velocities, and can be written as $\nu_1' = \nu_1 - \nu_1\bar{}$. $\nu_1\bar{}$ is the averaged velocity, computed with averaging of resolved velocity field.

It can be seen from the equations for $k_{res}$ and $\nu_1'$ that for $(\nu_1')^2$ calculation of the averaged velocity field is required. Therefore $Re_{tot}$ cannot be used as a switching criterion from the beginning of the simulation. To achieve successful simulation averaging should be done over time, when flow crosses domain at least once.

$$Re_{tot} \text{ as a switching criterion is used in a same way as } Re_k [2]:$$

$$Re_{tot} \leq C_{\text{switch}} \rightarrow \text{RANS mode},$$

$$Re_{tot} > C_{\text{switch}} \rightarrow \text{LES mode},$$

(10)

where the value of $C_{\text{switch}}$ can be used up to 60.

3.4. Modification of turbulent viscosity within the URANS region

By using total kinetic energy $k_{tot}$ instead of $k$ in the equations for LES models 3–6 and in the equations for URANS models written in Sections 3.2.1 and 3.2.2, we also used total turbulent viscosity $\nu_{SGS}^{\text{tot}}$ instead of the primary used modelled turbulent viscosity $\nu_{URANS}^{\text{tot}}$ [2]. Total turbulent viscosity $\nu_{SGS}^{\text{tot}}$ is the sum of the molecular and resolved turbulent kinetic energy: $\nu_{SGS}^{\text{tot}} = \nu_{URANS} + \nu_{SGS}^{\text{tot}}$. The resolved turbulent kinetic energy does not only depend on current velocity field but also on the difference between the current and averaged velocity fields. When simulating steady-state flow, the resolved turbulent viscosity is zero, $\nu_{SGS}^{\text{tot}} = 0$. Therefore, when simulating unsteady flow, the total turbulent viscosity $\nu_{SGS}^{\text{tot}}$ differs from the modelled turbulent viscosity $\nu_{URANS}$ within the range of unsteady character of the flow.

Too high values of $\nu_{URANS}$ could occur due to unsteady character of total turbulent kinetic energy $k_{tot}$ therefore they could be modified. We modified $\nu_{URANS}$, because $\nu_{SGS}^{\text{tot}}$ depends directly on flow.

The turbulent viscosity may be modified using a modification function $f$, Breuer et al. [2] defined $f$ from results of DNS turbulent.
channel flow as
\[
f = \begin{cases} 
0.3 \cdot R_{\text{tot}}, & 0 \leq R_{\text{tot}} \leq 1 \\
0.3, & 1 < R_{\text{tot}} \leq 12 \\
0.3 + (1.55 \cdot 10^{-4}(R_{\text{tot}} - 12)^2), & 12 < R_{\text{tot}} \leq 60
\end{cases}
\] (11)

Turbulent viscosity \( \nu_{\text{URANS}} \) can be modified with modification function \( f \)
\[
\nu_{\text{mod}}^{\text{URANS}} = \nu_{\text{URANS}}^{\text{mod}} \cdot (1 - f).
\] (12)

The above written modification function can only be used only for simulation, where \( R_{\text{tot}} \) is being used as the switching criterion.

3.5. URANS islands in LES region

During turbulent flow simulation using the LES/URANS hybrid model, URANS islands occur within the LES region. For investigating whether URANS islands adversely affect simulation, we have prescribed a condition which partly prevents URANS islands within the LES region. When the flow is unsteady and the interface between LES and URANS regions changes dynamically, we have focused only on one single node. We prescribe the condition, if a single node were to be in the URANS mode and all other boundary nodes in the LES mode, that the node also stays in the LES mode. It was used for both switching criteria, \( R_h \) and \( R_{\text{tot}} \).

3.6. Summary

In order to simplify the hybrid model’s labelling we allocated numbers to each hybrid model and its configurations, as shown in Table 1.

The models differ in turbulent model for URANS, turbulent kinetic energy, switching criterion between LES and URANS regions and modification function for URANS region. In models 1, 3 and 4 the turbulent model presented by Rodi is used and in models 2, 5 and 6 the turbulent model Rodi Higher Reynolds numbers is used. Models 1–2 are using turbulent kinetic energy \( k \) and models 3–6 are using total turbulent kinetic energy \( k_{\text{tot}} \). In models 1–2 the Reynolds number \( R_h \) is used for switching criterion and in models 3–4 the Reynolds number \( R_{\text{tot}} \) is used. The modification function is used in models 4 and 6.

4. Numerical method

A numerical algorithm based on subdomain BEM with velocity–vorticity formulation of Navier–Stokes equations was used for fluid flow simulation [27]. We divide the effective viscosity \( \nu_{\text{eff}} \) in BEM into the sum of constant and variable parts as
\[
\nu_{\text{m}} + \nu_{\text{v}} = \nu_{\text{eff}} = \nu_{0} + \tilde{\nu}.
\] (13)

We write thermal diffusivity in the same way:
\[
\alpha_{\text{m}} + \alpha_{\text{v}} = \alpha_{\text{eff}} = \alpha_{0} + \tilde{\alpha}.
\] (14)

4.1. Integral formulation of parabolic-diffusive fundamental solution

Kinetics equation (2) can be written as a non-homogeneous parabolic partial differential equation as
\[
\nu_{0} \frac{\partial^2 \omega}{\partial x_i \partial x_i} - \frac{\partial \omega}{\partial t} - \frac{1}{\rho_{0} c_{\text{p}}} \frac{\partial q_{y}}{\partial x_j} - e_{\nu} \frac{\partial^2 \nu}{\partial x_i \partial x_i} = 0
\] (15)

where the last three parts represent pseudo-body forces. Parabolic-diffusion fundamental solution \( u^* \) is defined as
\[
u^{\ast}(\xi, s, t, r) = \frac{1}{(4\pi \nu_{0} t)^{3/2}} e^{-r^2/4 \nu_{0} t},
\] (16)

where \((\xi, s, t)\) is a source point, \((s, t)\) the reference point within a domain, \( r \) the time step, and \( r \) the vector from source point \( \xi \) to reference point \( s \). The approximation on equations (17) implies a constant representation of the field variable during one time step. Using parabolic-diffusion fundamental solution \( u^* \) and
\[
u^* = \nu_{0} \int_{0}^{t_{n+1}} u^* dt, \quad \frac{\partial J^*}{\partial n} = \nu_{0} \int_{0}^{t_{n+1}} \frac{\partial u^*}{\partial x_i} dt, \quad \frac{\partial J^*}{\partial x_i} = \nu_{0} \int_{0}^{t_{n+1}} \frac{\partial u^*}{\partial x_i} dt,
\] (17)

we can write Eq. (15) in integral form for planar flow
\[
u(\xi, s, t, r) + \int_{f} \frac{\partial J^*}{\partial n} d\Gamma = \nu_{0} \int_{0}^{t_{n+1}} (\nu_{0} + \nu_{\text{v}}) \frac{\partial \omega}{\partial n} d\Gamma
\]

\[
+ \frac{1}{\nu_{0}} \int_{f} \left( \omega_{ij} - \nu_{\text{v}} \frac{\partial \omega}{\partial n} \right) \frac{\partial J^*}{\partial x_j} d\Gamma
\]

\[
+ e_{\nu} \frac{1}{\nu_{0}} \int_{f_{\text{t}}} \left( n \frac{\partial J^*}{\partial \Gamma} - \frac{\partial J^*}{\partial \partial \Gamma} \right) d\Omega
\]

\[
- e_{\nu} \frac{1}{\nu_{0}} \int_{f_{\text{t}}} \left( \frac{\partial J^*}{\partial \partial \Gamma} \right) d\Omega
\]

\[
+ \int_{f_{\text{t}}} \frac{\partial J^*}{\partial \partial \Gamma} d\Omega.
\] (18)

The same idea as used for the kinetics equation can be used for the kinematics equation. Kinematics equation (1) can be written in the same manner as Eq. (15). In regard to further simplification of the numerical algorithm, we choose the under-relaxation term of false transient time scheme \( \alpha \) to be equal to the viscosity term \( \nu_{0} \), and can write
\[
u_{0} \frac{\partial^2 v_{i}}{\partial x_i \partial x_i} - \frac{\partial v_{i}}{\partial t} + \nu_{0} e_{\nu} \frac{\partial \omega}{\partial x_i} = 0.
\] (19)

By considering (17) and (19), we may write kinematic equation (1) in integral form as
\[
u(\xi, s, t, r) + \int_{f} \frac{\partial J^*}{\partial n} d\Gamma
\]

\[
= \nu_{0} \int_{0}^{t_{n+1}} \frac{\partial J^*}{\partial n} d\Gamma
\]

\[
= \int_{f} \frac{\partial J^*}{\partial n} d\Gamma - \int_{f} \nu_{0} e_{\nu} \frac{\partial J^*}{\partial x_i} d\Gamma + \int_{f_{\text{t}}} \nu_{0} e_{\nu} \frac{\partial J^*}{\partial \Gamma} d\Omega + \int_{f_{\text{t}}} \nu_{0} e_{\nu} \frac{\partial J^*}{\partial \partial \Gamma} d\Omega.
\] (20)

Eq. (20) obtained for unknown domain velocity values can be written as
\[
u(\xi, s, t, r) + \int_{f} \frac{\partial J^*}{\partial n} d\Gamma
\]

\[
= \nu_{0} \int_{0}^{t_{n+1}} \frac{\partial J^*}{\partial n} d\Gamma - \int_{f} \nu_{0} e_{\nu} \frac{\partial J^*}{\partial x_i} d\Gamma + \int_{f_{\text{t}}} \nu_{0} e_{\nu} \frac{\partial J^*}{\partial \Gamma} d\Omega + \int_{f_{\text{t}}} \nu_{0} e_{\nu} \frac{\partial J^*}{\partial \partial \Gamma} d\Omega.
\] (21)

where \( n \) stands for unit normal and \( f \) for unit tangent.

Kinematics equation (1) is used also to determine boundary vorticity value, using the approach proposed by [26]. Eq. (20) rewritten for the solution of unknown boundary vorticity values can be written in vector form as
\[
u(\xi) \times \nu(\xi) + \nu(\xi) \times \int_{f} (\nabla U^* - \tilde{\nu}) \nabla U^* \, d\Gamma
\]
boundary elements: vectors of nodal values. Equation for kinetics (18) can be written as where \(c\) for computational cells. In the presented study we used quadratic parabolic-diffusion fundamental solution

\[
\begin{align*}
\frac{\partial \Phi}{\partial t} & = \nabla \cdot \left( \eta_{\nu} \frac{\nabla \Phi}{\partial n} \right) + \frac{1}{\eta_{\nu}} \sum_{c=1}^{\frac{E}{\Omega}} |g|^2 \left\{ (a_0 + u_T) \frac{\partial \nu}{\partial n} \right\}^p_f \\
& \quad - \frac{1}{\eta_{\nu}} \sum_{c=1}^{\frac{E}{\Omega}} |g|^2 \left\{ |T_{\nu}^2| + \frac{1}{\eta_{\nu}} \sum_{c=1}^{\frac{E}{\Omega}} |g|^2 \left\{ T_{\nu} - \nu \frac{\partial \nu}{\partial t} \right\}^p_f \right. \\
& \quad \left. + \sum_{c=1}^{\frac{E}{\Omega}} |g|^2 \left\{ \nu \frac{\partial \nu}{\partial t} \right\}^p_{f-1} \right\}.
\end{align*}
\]

According to boundary conditions we have assembled a system of equations written in matrix notation. We employed the numerical solver [24] for solving the system of equations.

5. Numerical algorithm

The implementation of a hybrid model within a numerical algorithm is shown step by step below. SGS and URANS turbulent viscosity, dissipation part and interface (step 6.1) are computed within the outer-loop for effective viscosity (step 6). Computed turbulent values \(\eta_{\nu}\) and \(D\) are further being used in the solver for turbulent kinetic energy \(k\) equation (step 6.4) and in next time-step (step 1) for governing equations for fluid flow (steps 3–5).

1. Time loop
2. Global iteration loop
3. Kinetics loop
3.1 kinematics, solve boundary vorticity \(T\)
3.2 kinematics, solve internal velocity \(v\)
3.3 check convergence \(T\), \(v\) if error is greater than predefined convergence criterion \(\epsilon_{m}\) go to 3.1
4. kinetics, solve energy equation
5. kinetics, internal vorticity \(\omega\)
6. outer loop for effective viscosity \(\nu_{\text{eff}}\)
6.1 inner iteration loop for effective viscosity \(\nu_{\text{eff}}\)
6.2 turbulence model, calculate turbulence viscosity and dissipation part for sub-grid model \(\nu_{SGS}, D_{SGS}\). Eqs. (6) and for URANS model \(\nu_{URANS}, D_{URANS}\), Sections 3.2.1 and 3.2.2, interface between LES and URANS region. Section 3.3, define \(D\) and \(\nu\) for further time steps 6.5 and 1.
6.3 inner iteration loop for turbulent kinetic energy \(k\)
6.4 kinetics, solve turbulent kinetic energy equation \(k\)
6.5 check convergence for \(k\), if error is greater than \(\epsilon_k\) go to 6.3
6.6 check convergence for \(\nu_{\text{eff}}\) if error is greater than \(\epsilon_{\text{eff}}\) go to 6.2
6.7 check convergence for \(\nu_{\text{eff}}\) if error is greater than \(\epsilon_{\text{eff}}\) go to 6.1
7. time loop, if simulation not finished go to 1.
8. end of computation, write results

The convergence criteria used in the numerical algorithm were \(\epsilon_k = 10^{-4}\), \(\epsilon_{\text{eff}} = 5 \times 10^{-6}\), \(\epsilon_l = 10^{-3}\), and \(\epsilon_m = 10^{-4}\). We used under-relaxations of all equations using values between 0.1 and 0.3.

6. Simulation of natural convection flow within a square cavity

We tested our numerical algorithm by simulating an unsteady turbulent natural convection flow within a square cavity. One of the first authors who studied such flows was Davies [5]. He published results regarding the simulation of natural convection flow within a square cavity up to Rayleigh number \(Ra = 10^4\), and proposed natural convection flow within the square cavity as a benchmark case for solving non-isothermal fluid problems. Further research was done by Lo et al. [11], who used the velocity–vorticity formulation of Navier–Stokes equations for
natural convection flow simulation up to $Ra = 10^9$. Sajjadi et al. [20] investigated natural convection flow within a square cavity employing Large Eddy Simulation and the Lattice Boltzmann method for Rayleigh numbers from $Ra = 10^6$ to $Ra = 10^9$. Natural convection flow within the square cavity is steady up to Rayleigh number $Ra = 10^7$. At $Ra = 10^8$ it becomes unsteady, but not yet turbulent, the flow is not chaotic. When the flow reaches Rayleigh number $Ra = 10^8$, the flow is unsteady, chaotic, and turbulent. The boundary conditions are defined in Fig. 1 (left): temperature $T_b = 0.5$ on the uniformly heated left wall, adiabatic boundary condition $q = 0$ on the upper and lower walls, temperature $T_c = -0.5$ on the uniformly cooled right wall, no slip velocity $\vec{v} = 0$ boundary conditions on all four walls, and gravity $\vec{g}$ in the opposite direction of coordinate $y$. The problem is defined by the Rayleigh number. The Rayleigh number and the Prandtl number, which is $Pr = \frac{\nu}{\alpha}$, can be defined as

$$Ra = \frac{g \cdot \beta \cdot \Delta T \cdot L^3}{\nu \cdot \alpha}, \quad Pr = \frac{\nu}{\alpha}$$

We assumed that the medium in the cavity is air with the following fluid properties: kinematic viscosity $\nu = 17.6 \cdot 10^{-6}$ m$^2$/s, thermal volumetric expansion coefficients $\beta = 1/313.15$ K$^{-1}$ and thermal conductivity $\lambda = 0.0265$ W/mK. The height and width of a cavity is $L = 0.5$ m. Physical temperature difference between the heated and cooled walls is $\Delta T = 111.4$ K at $Ra = 10^9$.

During simulation, we used computational mesh with $80 \times 80$ domain elements and $161 \times 161$ nodes. The mesh was compressed in the normal wall direction in both coordinates with the geometrical series having the ratio between the longest and shortest elements $R = 10$.

The results of simulating natural convection flow are evaluated using the averaged Nusselt number. The Nusselt number measures the heat flux through a solid wall and can be defined as

$$Nu = \frac{1}{H} \int_0^H \frac{\partial T}{\partial y} dy.$$  \hspace{1cm} (29)

7. Results

7.1. Laminar natural convection at $Ra = 10^3$–$10^8$

Before employing the LES/URANS hybrid model and choosing appropriate computational mesh, we validated our numerical algorithm on laminar simulations of natural convection flow for Rayleigh numbers from $Ra = 10^3$ to $Ra = 10^8$. Simulations up to $Ra = 10^7$ were steady and simulation with $Ra = 10^8$ unsteady. For numerical algorithm validation we used three different computational meshes: $40 \times 40 \times 10$, $80 \times 80 \times 10$, and $120 \times 120 \times 10$; which meant $81 \times 81$, $161 \times 161$, and $241 \times 241$ nodes. Simulation results for the lower and middle Rayleigh numbers are shown in Table 2, and are compared with the results from other authors. Simulations with computational mesh $40 \times 40 \times 10$ up to $Ra = 10^8$ provided comparable results as simulations using $80 \times 80 \times 10$ and $120 \times 120 \times 10$. Mesh $40 \times 40 \times 10$ was too coarse for natural convection flow simulation at $Ra = 10^9$, high temperature gradients and a thin boundary layer can be seen in further results. It should be noticed that, for mesh $40 \times 40 \times 10$, the simulations for $Ra = 10^7$ and $Ra = 10^8$ were unsteady, results in brackets ($\bar{)}$ in Table 2. We concluded from mesh analysis using DNS for low and middle Rayleigh numbers, that in further simulations mesh $80 \times 80 \times 10$ would be used. It can be seen that the values of the Nusselt number are within the same range; therefore we concluded that a numerical algorithm is appropriate for further simulations using turbulent models.

![Fig. 1. Boundary conditions for test case simulation of natural convection flow within square cavity (left). Mesh used in simulation with $80 \times 80$ computational cells and ratio between longest and shortest elements $R = 10$ (right).](image)

Table 2

Nusselt number for natural convection flow simulation using DNS for Rayleigh number from $Ra = 10^3$ to $Ra = 10^9$. Values in brackets ($\bar{)}$) display the averaged results for unsteady laminar simulation and the values in square brackets, the results when using turbulent models.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$10^3$</td>
<td>1.114</td>
<td>1.118</td>
<td>1.118</td>
<td>1.118</td>
<td>1.1178</td>
<td>1.1173</td>
<td>1.1174</td>
<td></td>
</tr>
<tr>
<td>$10^4$</td>
<td>2.245</td>
<td>2.243</td>
<td>2.224</td>
<td>2.224</td>
<td>2.2438</td>
<td>2.2434</td>
<td>2.2438</td>
<td></td>
</tr>
<tr>
<td>$10^6$</td>
<td>8.806</td>
<td>8.8</td>
<td>8.823</td>
<td>8.7</td>
<td>8.8231</td>
<td>8.8222</td>
<td>8.8200</td>
<td></td>
</tr>
<tr>
<td>$10^7$</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>16.79</td>
<td>17.2</td>
<td>(16.518)</td>
<td></td>
</tr>
<tr>
<td>$10^8$</td>
<td>[32.3] (30.1)</td>
<td>(30.506)</td>
<td>[31.2]</td>
<td>(30.119)</td>
<td>(30.15)</td>
<td>(30.16)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Fig. 2 shows the temperature field for \( Ra = 10^7 \) (left) and \( Ra = 10^8 \) (right). It can be seen from Fig. 2 that the temperature isolines are in the middle of the cavity almost parallel, and temperature stratification occurs. Natural convection flow up to Rayleigh number \( Ra = 10^7 \) is stationary, but at \( Ra = 10^8 \) flow becomes unsteady, but not yet chaotic and turbulent. We used DNS for simulations up to \( Ra = 10^8 \), the time-step used in the simulation at \( Ra = 10^8 \) was \( \Delta t = 10^{-5} \).

7.2. Turbulent natural convection at \( Ra = 10^9 \)

Simulations of turbulent natural convection flow were performed using 6 different hybrid LES/URANS turbulent models. Time step used in the simulation at \( Ra = 10^8 \) was \( \Delta t = 5 \times 10^{-7} \). Time series for calculating the averaged Nusselt number was approximately 6000 time steps for hybrid models 1 and 2, approximately 11,000 for hybrid models 3 and 4, and approximately 8500 for hybrid models 5 and 6.

Fig. 4 shows temperature–vorticity phase portraits for \( Ra = 10^9 \) using computational mesh \( 80 \times 80 \times 10 \) and different hybrid models, 1–6. It can be seen that the nature of the flow is turbulent, unsteady, and chaotic. In natural convection flow, transition to turbulent flow occurs between \( Ra = 10^7 \) and \( 10^8 \). The unsteady and chaotic characteristics of natural convection flow can be seen in Fig. 6, that shows the temperature and the vorticity field for \( Ra = 10^9 \) using hybrid models 1 and 3. The main force acting on the fluid in the natural convection flow is buoyancy due to temperature difference and gravity. As the vertical walls are being heated and cooled, fluid accelerates when passing by vertical walls. Although the flow is turbulent near the vertical walls, the majority of eddies are formed in the corners. In the figures it can also be seen that the boundary layer near the vertical walls is thin and the temperature gradients are high, therefore mesh should be compressed towards the walls. Fig. 3 (right) shows temperature profiles for Rayleigh number \( Ra = 10^9 \) at height \( y = L/2 \) and length \( x = L/2 \), using hybrid models from 1 to 6. The biggest difference between the temperature profiles for \( Ra = 10^8 \) and \( Ra = 10^9 \) is in the vertical profile in length \( x = L/2 \). Although the profiles for \( Ra = 10^9 \) show the same trend, it can be seen that the profile for hybrid model 1 deviates slightly from those profiles using other hybrid models. Fig. 5 shows the temperature time series power spectra at points A (0.05061, 0.05061) and B (0.09857, 0.09857) for different hybrid models. A power spectrum was calculated using fast Fourier Transform. Power spectra show the frequencies of eddies forming oscillations and consequently the presence of eddies of the different sizes. It can be seen that the widest frequency band occurs using models 1 and 2; the presence of smaller and larger eddies is more uniform. We noticed the power decay with the frequency, which was also noticed by other researchers [15].

When comparing the temperature and vorticity field in Fig. 6 with the LES/URANS regions in Fig. 8, we can see that URANS was used in near walls regions where attached eddies occurred and LES was used where detached eddies occurred. It can be seen from the time trace of the Nusselt number (Fig. 7) and the averaged Nusselt number values (Table 3) that the modification function has an effect on the flow. As turbulent viscosity \( \nu_t \) and heat conductivity are in linear connection with the turbulent Prandtl number \( Pr_t \), no clear trend could be done.
whether the impact is due to lower turbulent viscosity or lower heat conductivity within URANS region.

For $Ra = 10^9$ the value of switching criterion was $C_{\text{switch}} = 30$ for hybrid models 1 and 2, and $C_{\text{switch}} = 60$ for hybrid models 3–6. Fig. 8 shows LES and URANS zones of hybrid model in natural convection simulation at $Ra = 10^9$. Graphs in Fig. 9 show a ratio of the LES region area according to the whole simulation area. It can be seen from Figs. 8 and 9 that models with switching criterion $Re_{\text{tot}}$ 3–6 have more dynamic interfaces than models 1 and 2, where $Re_k$ is used for switching criterion. The difference in
interface dynamics occurs when defining the quantity used for the switching criterion. Although $R_{\text{eq}}$ is a dynamic quantity, it only depends on the current modelled turbulent kinetic energy $k$. $R_{\text{e tot}}$ depends on the current modelled turbulent kinetic energy $k$ and on the difference between the averaged and current solved velocity fields. The graph in Fig. 7 (left) shows the Nusselt number for $Ra = 10^9$ for all hybrid models. It can be seen that the smallest amplitude of the Nusselt number is by using hybrid models 1 and 2, and the biggest using hybrid model 4. The results obtained using hybrid models 1 and 2 are damper in comparison with the results obtained using models 3–6.

The averaged Nusselt number results for $Ra = 10^9$ are summarized in Table 3 and compared with the results from other researchers. Barakos et al. [1] used DNS and two-equation URANS model $k-\epsilon$ for natural convection flow simulation, averaged Nusselt numbers for $Ra = 10^9$ were 54.4 using DNS and 60.1 using $k-\epsilon$. Dixit and Babu [6] used the lattice Boltzmann method, averaged Nusselt number for $Ra = 10^9$ was 57.35. It can be seen from averaged Nusselt number, that the results obtained with the LES/URANS hybrid model are within same range. From graphs with Nusselt number 7 and from the averaged Nusselt number data (Table 3) it can be seen that different URANS models, switching criteria and the use of a modification function do affect the simulation results. URANS islands did not adversely impact on hybrid models 1 and 2, where the Reynolds number based on turbulent kinetic energy $R_{\text{eq}}$ is used as the switching

![Fig. 6. Temperature (left) and vorticity (right) field at $Ra = 10^9$ using hybrid model 1 in time step 13843 (top) and hybrid model 3 in time step 29719 (bottom).](image1)

![Fig. 7. Nusselt number for Rayleigh number $Ra = 10^9$ using computational mesh $80 \times 80$ in $R_1$. In the left of the figure are results for hybrid model 1 (---) and hybrid model 2 (----), in the middle figure for hybrid model 3 (---) and hybrid model 4 (----) and in the right figure for hybrid model 5 (---), and hybrid model 6 (----).](image2)

<table>
<thead>
<tr>
<th>Hybrid model</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>Barakos et al. [1]</th>
<th>Dixit and Babu [6]</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Ra = 10^9$</td>
<td>56.0</td>
<td>56.5</td>
<td>57.7</td>
<td>56.2</td>
<td>52.6</td>
<td>52.9</td>
<td>(54.4) 60.1</td>
<td>57.35</td>
</tr>
</tbody>
</table>

Table 3

Nusselt numbers for natural convection flow simulation using different turbulent LES/URANS hybrid models.
criteria. On the other hand, URANS islands do have a slight impact on simulations using hybrid models where the Reynolds number based on total turbulent kinetic energy $Re_{tot}$ was used, that means hybrid models 3–6. URANS islands do not affect the temperature field directly, but they do affect the turbulent quantities, turbulent viscosity $\nu_t$ and turbulent heat conductivity $\alpha_t$. Because $\nu_t$ and $\alpha_t$ are being further used in turbulent kinetic energy $k$ transport equation computation and fluid flow computation, it significantly affects convergence.

8. Conclusions

The developed LES/URANS hybrid turbulent model was, in combination with BEM based algorithm and velocity–vorticity formulation of governing equations, successfully used for solving unsteady non-isothermal turbulent phenomena. The numerical algorithm was tested using natural convection flow within a square cavity. We used a direct simulation for Rayleigh numbers up to $Ra = 10^6$. Fluid flow up to $Ra = 10^7$ is steady, at $Ra = 10^8$ the flow becomes unsteady, but still periodic and non-chaotic. Transition to turbulent flow occurs between $Ra = 10^5$ and $10^6$. It can be seen from the temperature and vorticity field, and the vorticity–temperature phase diagrams, that flow becomes unsteady and chaotic. A unified one-equation LES/URANS hybrid turbulent model, that is based on transport equation for turbulent kinetic energy $k$, was used for developed turbulent flow. The interface between LES and URANS is defined dynamically during simulation. For switching criterion we employed the Reynolds number based on turbulent kinetic energy $Re_k$ and the Reynolds number based on total kinetic energy $Re_{tot}$. In further simulations, we used a modification function for the URANS region. The results obtained with the LES/URANS hybrid model for $Ra = 10^9$ are in good agreement with the results from Barakos et al. [1] and results from Dixit and Babu [6].

References


