Large Eddy Simulation of turbulence induced secondary flows in stationary and rotating straight square ducts

W. Sudjai¹, V. Juntasaro¹* and V. Juttijudata²

¹Department of Mechanical Engineering, Faculty of Engineering, Kasetsart University, Bangkok, Bangkok 10900, Thailand
²Department of Aerospace Engineering, Faculty of Engineering, Kasetsart University, Bangkok, Bangkok 10900, Thailand

* Corresponding Author: varangrat.j@ku.th

Abstract. The accuracy of predicting turbulence induced secondary flows is crucially important in many industrial applications such as turbine blade internal cooling passages in a gas turbine and fuel rod bundles in a nuclear reactor. A straight square duct is popularly used to reveal the characteristic of turbulence induced secondary flows which consists of two counter rotating vortices distributed in each corner of the duct. For a rotating duct, the flow can be divided into the pressure side and the suction side. The turbulence induced secondary flows are converted to the Coriolis force driven two large circulations with a pair of additional vortices on the pressure wall due to the rotational effect. In this paper, the Large Eddy Simulation (LES) of turbulence induced secondary flows in a straight square duct is performed using the ANSYS FLUENT CFD software. A dynamic kinetic energy subgrid-scale model is used to describe the three-dimensional incompressible turbulent flows in the stationary and the rotating straight square ducts. The Reynolds number based on the friction velocity and the hydraulic diameter is 300 with the various rotation numbers for the rotating cases. The flow is assumed fully developed by imposing the constant pressure gradient in the streamwise direction. For the rotating cases, the rotational axis is placed perpendicular to the streamwise direction. The simulation results on the secondary flows and the turbulent statistics are found to be in good agreement with the available Direct Numerical Simulation (DNS) data. Finally, the details of the Coriolis effects are discussed.

1. Introduction

The understanding of the characteristic of turbulence induced secondary flows and its change when involved with the Coriolis effects is crucially important since the flows appear in many industrial applications such as turbine blade internal cooling passages in a gas turbine and fuel rod bundles in a nuclear reactor. In order to reveal the mechanism of the flows, the simple straight square duct is popularly used as the compatible observation case of these phenomena.

In the literature, the turbulent flows through a stationary straight square duct exposed the characteristic of turbulence induced secondary flows which consists of two counter rotating vortices distributed in each corner of the duct. The flows is relatively weak as about 2 per cent of the bulk velocity but it has significant effects on the momentum transfer and the wall shear stress distribution [1-5]. Gavrilakis [2] provided the useful database of the mean velocity and the turbulence statistics of the fully developed flows in a stationary straight square duct at the friction Reynolds number of 300 by using Direct Numerical Simulation (DNS). Thereafter, Huser [3] performed DNS on the stationary duct at the
friction Reynolds number of 600, and reported the mechanism of the turbulence induced secondary flows. Huser [3] concluded that the reduction of mean shear stress in the corner bisector allows the high momentum fluid flows from the center region of the duct towards the corners. Consequently, the secondary flows penetrate more toward the corners as the Reynolds number increases [5, 6]. Recently, the highest Reynolds number of the turbulent flows simulation in the stationary duct of DNS has been performed up to 1200 of the friction Reynolds number [6], by the way, the widest range of Large Eddy Simulation (LES) of the flows has been carried out from 300 to 10550 of the friction Reynolds number [5].

Coriolis effects on the turbulence induced secondary flows was presented in the turbulent flows through a rotating straight square duct. In the rotating duct, the flows can be divided into a pressure side and a suction side. As the rotation number increases, the turbulence is enhanced on the pressure side whereas the flows is stabilized on the suction side [7-11]. The rotational effects convert the turbulence induced secondary flows of the stationary duct into the Coriolis force driven two large circulations with a pair of additional vortices on the pressure side. In different rotation numbers, these additional vortices were appeared and disappeared with fixed reattachment points and various positions of separation points on the pressure side [10]. Taylor-Proudman zone, where the streamwise velocity along the spanwise direction is constant, is located at the central region of the duct.

The Taylor-Proudman zone is expanded in the spanwise direction as the rotation number increases [10, 11]. The present of Ekman layers on the lateral walls create suction regions on the pressure side close to the corners. In addition, the positions of the suction regions behave as fixed boundary of the additional vortices since it does slightly change due to the suppression of turbulence near the corners [10]. The Coriolis effects causes the additional resistance on the streamwise mean momentum [11]. Therefore, bulk velocity and flow rate in the duct decrease as the rotation number increases [7-11]. Recently, at the low friction Reynolds number of 300, Dai et al. [10] performed DNS of the turbulent flows in the rotating duct with the range of the friction rotation number based on the mean friction velocity and the full duct width from 0 to 40. Moreover, Fang et al. [11] carried out DNS of the turbulent flow in the rotating duct at the friction Reynolds number based on the mean friction velocity and the half duct width of 150 with the wide range of the friction rotation number based on the mean friction velocity and the half duct width from 0 to 27. Pallares and co-workers [7-9] performed LES with the localized one-equation dynamic subgrid scale model [12] to study the rotation effects on the secondary flows and heat transfer in the square duct at different rotation numbers.

In this paper, ANSYS FLUENT 18.1 commercial software was used to reveal the characteristic of turbulence induced secondary flows in stationary and rotating straight square ducts. LES with a dynamic turbulent kinetic energy subgrid scale model [12] was carried out to describe the three-dimensional incompressible fully developed turbulent flows in the ducts. The Reynolds number based on the friction velocity and the hydraulic diameter is 300. For the rotating case, the two different friction rotation numbers of 1.3 and 5 were considered. The former is where the turbulence induced secondary flows is suppressed and the latter is where the additional vortices are located on the pressure side [8]. Moreover, the accuracy of the software has been reported.

2. Numerical simulation

2.1. Physical model and boundary conditions

The coordinate system and the computational domain of the straight square duct are shown on figure 1. The coordinate system (x, y, z) represent streamwise, normal, and spanwise direction, respectively. The velocity components (u, v, w) denote velocity component in x, y and z direction, respectively. The rotation axis is placed at the center of the computational domain with the spanwise rotation direction. The size of a cross sectional area of the duct is D X D where D is the hydraulic diameter. The streamwise length of 6.4D is used in this work. In literature, the simulation of force convection heat transfer in a rotating square duct at high rotation numbers using a dynamical localized subgrid scale model [12] with the shortest of the streamwise length computational domain of 6D was carried out by Pallares et al. [9].
Therefore, the streamwise length computational domain in this work is sufficiently long enough to capture the effects of the secondary flows. No-slip conditions is imposed at each walls. The friction Reynolds number is defined as $Re_\tau = u_\tau D/\nu$ and the friction rotation number is defined as $Ro_\tau = 2\Omega D/u_\tau$ where $u_\tau$ denotes the friction velocity, $\nu$ denotes the kinematic viscosity and $\Omega$ denotes the rotational angular velocity. In order to observe the effects of Coriolis force in rotating square ducts, the friction Reynolds number, $Re_\tau$, is fixed at 300, and the two different friction rotation numbers, $Ro_\tau$, of 1.3 and 5 are considered. The fully developed flow is assumed by imposing periodic boundary condition with the constant pressure gradient, $dP/dx = -4$, along the x-axis.

![Figure 1. Computational domain](image)

### 2.2. Numerical methods

In Large Eddy Simulation (LES), turbulent flows are decomposed into large eddies, which contains the most of energy, and universal isotropic small eddies by applying the filtering operation. The large eddies are directly resolved while the small eddies are modeled.

The filtering incompressible Navier-Stoke equations are obtained for the resolved large eddies. Thus, the governing equations are

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho \bar{u}_i) = 0 \tag{1}
\]

and

\[
\frac{\partial}{\partial t} (\rho \bar{u}_i) + \frac{\partial}{\partial x_j} (\rho \bar{u}_i \bar{u}_j) = \frac{\partial}{\partial x_j} (\sigma_{ij}) - \frac{\partial \bar{p}}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}
\]

where $\bar{u}_i$ is the resolved velocity field, $\bar{p}$ is the resolved pressure, $\rho$ is the density and $t$ is time, and $\sigma_{ij}$ is the stress tensor due to molecular viscosity defined by

\[
\sigma_{ij} \equiv \left[ \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \right] - \frac{2}{3} \mu \frac{\partial \bar{u}_l}{\partial x_l} \delta_{ij} \tag{3}
\]

and $\tau_{ij}$ is the subgrid scale stress defined by

\[
\tau_{ij} \equiv \rho \bar{u}_i \bar{u}_j - \rho \bar{u}_i \bar{u}_j \tag{4}
\]
The unresolved small eddies in the subgrid scale stress must be modeled. In this paper, the LES with a dynamic kinetic energy subgrid scale model proposed by Kim and Menon [12] is implemented using the ANSYS FLUENT 18.1 CFD software.

The transport equation of the subgrid scale kinetic energy, $k_{sgs}$, is obtained as

$$\rho \frac{\partial k_{sgs}}{\partial t} + \rho \frac{\partial \bar{u}_i k_{sgs}}{\partial x_j} = -\tau_{ij} \frac{\partial \bar{u}_i}{\partial x_j} - C_e \rho \frac{k_{sgs}^{3/2}}{\Delta_f} + \frac{\partial}{\partial x_j} \left( \mu_t \frac{\partial k_{sgs}}{\partial x_j} \right)$$

(5)

where $C_e$ is a dynamically adjustable coefficient.

The subgrid scale kinetic energy is obtained by solving the transport equation of Eq. (5)

$$k_{sgs} = \frac{1}{2} \left( \bar{u}_k^2 - \bar{u}_k^2 \right)$$

(6)

The subgrid scale eddy viscosity, $\mu_t$, is obtained by computing the $k_{sgs}$ as

$$\mu_t = C_k \rho k_{sgs}^{1/2} \Delta_f$$

(7)

where $C_k$ is a dynamically adjustable coefficient and $\Delta_f$ is the filter size computed from $\Delta_f \equiv V^{1/3}$. Therefore, the subgrid scale stress, $\tau_{ij}$, is modeled as follows,

$$\tau_{ij} - \frac{2}{3} \rho k_{sgs} \delta_{ij} = -2C_k \rho k_{sgs}^{1/2} \Delta_f \tilde{S}_{ij}$$

(8)

Pallares and co-workers [7-9] coded the localized dynamic subgrid scale model or also called the dynamic kinetic energy subgrid scale model by themselves. Their codes were done successfully in the simulations of the turbulent flow and heat transfer in a square duct with and without rotation. On the other hand, this paper used the commercial software ANSYS FLUENT 18.1 to describe the physics of the turbulent induced secondary flows in stationary and rotating straight square ducts and also to ensure that the software is capable to characterize the secondary flows and the effects of Coriolis force due to the rotation in the ducts.

ANSYS Meshing was used to build up the proper computational mesh in order to capture the complexity of the secondary flows in the ducts. Multizone method was used to generate the hexahedral mesh. Uniform mesh was created, in order to capture the streamwise streaks of turbulent structure, by using the edge sizing with selected, no bias, number of divisions. In the cross sectional area of the ducts, the edge sizing with the selected number of divisions with the smooth transition bias growth rate was used along the perimeter of the cross sectional area to generate the proper mesh which can capture the secondary flows. Two types of computational mesh were considered. A coarse mesh type has 66 X 40 X 40 number of divisions along the edge in x, y and z direction, respectively, with the bias growth rate of 1.2 along the cross sectional area. This corresponds to a uniform grid spacing of $\Delta x^+ \approx 29.09$ along the streamwise direction and $(\Delta y^+)_\text{min} = (\Delta z^+)_\text{min} = 0.8$ and $(\Delta y^+)_\text{max} = (\Delta z^+)_\text{max} \approx 25.55$ on the cross sectional area. For a fine mesh type, the computational domain was divided into 80 X 80 X 80 number of divisions with the bias growth rate of 1.0678 along the cross sectional area. The corresponding grid spacing are a uniform grid spacing of $\Delta x^+ \approx 24$ along the streamwise direction and $(\Delta y^+)_\text{min} = (\Delta z^+)_\text{min} = 0.8$ and $(\Delta y^+)_\text{max} = (\Delta z^+)_\text{max} \approx 10.33$ on the cross sectional area.

Large Eddy Simulations of both the stationary and rotating straight square ducts were started from precursor solutions of RANS (Reynolds Averaged Navier-Stoke) with a superimposed synthetic turbulence on the mean flow at the individual rotation rates ($Ro_\tau = 0, 1.3$ and 5). The precursor RANS solutions with the perturbations were used as initial condition in each coarse mesh LES cases. Only the stationary duct case was computed up to the fine mesh due to the computational resources. The initial
condition of the fine mesh LES stationary duct case was obtained from the coarse mesh LES case at $Ro = 0$. The sampling procedure was started after the solution reached the statistically steady state to obtain the averaged velocity field and the turbulent intensities. The sampling rate does not exceed $0.1 D/\nu$. For the fine mesh stationary duct case, the mean flow field was obtained over $280 D/\nu$ period of time. Likewise, the mean flow field of the coarse mesh rotating duct cases were obtained over $270 D/\nu$ and $280 D/\nu$ for the friction rotating number of 1.3 and 5, respectively. Time steps were chosen to keep the maximum CFL number below 0.6.

ANSYS FLUENT 18.1’s pressure based solver was used to obtain the solutions. For pressure-velocity coupling, the Fractional Step Method was used. For spatial discretization scheme, least squares cell based was used for gradient scheme, second order for pressure scheme, central differencing for momentum and subgrid kinetic energy scheme. Second order implicit was used for transient formulation scheme. Non-iterative time advancement was enabled.

3. Results and discussions

The results of the present ANSYS FLUENT 18.1’s LES with a dynamic kinetic energy subgrid scale model proposed by Kim and Menon [12] of the turbulent flows in the stationary and rotating straight square ducts are presented comparing with the available LES and DNS databases. As shown on table 1, the simulated Reynolds number and rotation number based on the bulk velocity, $U_b$, and the hydraulic diameter, D, at different rotation rates are agree reasonable well with the available databases with the different about 1.5 percent.

3.1. Stationary straight square duct

For the stationary case, the present ANSYS FLUENT 18.1’s LES with a dynamic kinetic energy subgrid scale model proposed by Kim and Menon [12] has been compared with the available database of DNS of Gavrilakis [2] and LES of Yao et al. [5]. In addition, Yao et al. [5] performed LES of turbulent flows through a stationary straight square duct from 300 to 10550 of the friction Reynolds numbers with a dynamic Smagorinsky subgrid scale model proposed by Germano et al. [13]. Thus, the LES with another dynamic subgrid scale model [13] at the friction Reynolds number of 300 was chosen for the comparison. The comparisons of the stationary straight square duct case were presented in figure 2-8. The $y/D$ denotes the dimensionless distance along normal direction from -0.5 to 0.5 and the $z/D$ denotes the dimensionless distance along the spanwise direction from -0.5 to 0.5.

3.1.1. Mean flow field and turbulent intensities. The mean flow field was obtained over a time period of $280 D/\nu$, with the fine mesh. The mean velocity profiles were normalized by the mean streamwise velocity at the centerline of the duct, $u_0$ since the mean centerline streamwise velocity scales were available from the reference databases. Figure 2 shows the normalized mean streamwise velocity profiles at five locations of the straight square duct, i.e. $z/D = -0.45, -0.35, -0.25, -0.15, and 0$. The present LES mean streamwise velocity profiles have better agreement with DNS of Gavrilakis [2] than the other LES [5] at all five locations of the duct. Since, solving the flow field with the one transport equation of the dynamic kinetic energy subgrid scale model [12] would give more realistic characteristic of the flows than just only solving the flow field with the algebraic equation of the dynamic Smagorinsky subgrid scale model [13].

On the other hand, the secondary flows were obtained with the good agreement between the present LES and both simulation databases of normalized mean spanwise velocity at five locations of the duct, i.e. $z/D = -0.42, -0.35, -0.25, -0.15, and -0.1$, as shown on figure 3. Furthermore, the maximum mean secondary velocity is about 2 per cent of the bulk streamwise velocity.
Table 1. Comparison between the simulated bulk Reynolds number and bulk rotation number and the available databases with the three different friction rotation number of 0, 1.3 and 5 at the fixed friction Reynolds number of 300.

| Authors         | $Ro_{\tau} = 0$ | $Ro_{\tau} = 1.3$ | $Ro_{\tau} = 5$ |
|-----------------|-----------------|-------------------|-----------------|
|                 | $Re_b$ | $Ro_b$ | $Re_b$ | $Ro_b$ | $Re_b$ | $Ro_b$ |
| Present LES     | 4480.48 | 0     | 3974.833 | 0.098117 | 3066.302 | 0.489189 |
| Gavrilakis [2]  | 4410   | 0     | -     | -     | -     | -     |
| Yao et al. [5]  | 4410   | 0     | -     | -     | -     | -     |
| Pallares et al. [9] | - | - | -     | -     | 3080  | 0.49  |
| Dai et al. [10] | 4418   | 0     | 3924  | 0.099 | 3124  | 0.48  |

Figure 2. Comparison of the present LES mean streamwise velocity profiles with DNS of Gavrilakis [2] and LES of Yao et al. [5] at (a) $z/D = -0.45$; (b) $z/D = -0.35$; (c) $z/D = -0.25$; (d) $z/D = -0.15$; (e) $z/D = 0$. 


Figure 3. Comparison of the present LES mean spanwise velocity profiles with DNS of Gavrilakis [2] and LES of Yao et al. [5] at (a) z/D = -0.42; (b) z/D = -0.35; (c) z/D = -0.25; (d) z/D = -0.15; (e) z/D = -0.1.

The velocity intensities were normalized by the mean streamwise velocity at the centerline of the duct, \( u_0 \). Figure 4 shows the comparison of \( u_{rms}/u_0 \) at five locations of the duct. The good agreement between the present LES and both simulation databases were obtained. As shown on figure 4, the local minimum of \( u_{rms}/u_0 \) are located along the corner bisector and which indicate the effect of the secondary flows. The highest value of \( u_{rms}/u_0 \) is placed near the wall at the middle of the duct. However, the values decrease as its location approach to the corner.

Figure 6 displays the mean secondary velocity vectors and the dimensionless mean streamwise velocity contours and the mean secondary velocity streamline. As shown on figure 6(a) and figure 6(b), the turbulence induced secondary flows consists of two counter rotating vortices distributed in each corner of the duct. The high dimensionless mean streamwise velocity, \( u^+ = u/u_\tau \), zone placed at each...
corner of the duct represent the effect of the turbulence induced secondary flows. The secondary flows transfer the high momentum fluid from the center region of the duct towards the corners along the corner bisectors where the local minimum of \( u_{rms}/u_0 \) are located, as mentioned in figure 4.

Figure 7 represents the distribution of wall shear stress, \( \tau_w \), along the duct wall. The wall shear stress distribution of the present LES was compared with only DNS of Gavitakis [2] since there were no available wall shear stress distribution results of any LES. Both results show two local maximum of wall shear stress with some different of local maximum wall shear stress at the middle of the duct due to the sampling procedure. Near the corner, the present LES wall shear stress rapidly rises up to the local maximum at the dimensionless distance along the duct wall of about -0.35 due to the presence of the secondary flows which transfer the high momentum fluid from the center region of the duct towards the corners. Furthermore, the occurrence of the local maximum wall shear stress at the middle of the duct wall represents the Reynolds number effect [2].

![Figure 4](image.png)

**Figure 4.** Comparison of \( u_{rms}/u_0 \) profiles of the present LES with DNS of Gavitakis [2] and LES of Yao et al. [5] at (a) \( z/D = -0.45 \); (b) \( z/D = -0.35 \); (c) \( z/D = -0.25 \); (d) \( z/D = -0.15 \); (e) \( z/D = 0 \).
3.1.2. Turbulence structures. Figure 8(a) displays turbulence structures with turbulence intensity contours for the flows inside the stationary duct ($\text{Ro}_x = 0$). The Q-criterion is used to expose the turbulence structures. The turbulence structures are thoroughly distributed inside the duct due to the effects of the secondary flows. As shown on figure 8(a), the turbulence structures stretched along the duct in the streamwise direction. In addition, the turbulence intensity, $I$, is defined as the ratio of the root mean square of the velocity fluctuations to the mean flow velocity. Near the wall, the turbulence intensity is weak due to the wall damping effects.

3.2. Rotating straight square duct
For the rotating case, the present ANSYS FLUENT 18.1’s LES with a dynamic kinetic energy subgrid scale model proposed by Kim and Menon [12] has been compared with the available databases of DNS of Dai et al. [10] and DNS of Fang et al. [11]. The comparison of the rotating straight square ducts at two different friction rotation numbers of 1.3 and 5 with the constant friction Reynolds number of 300.
were presented in figures 6, 8, 9, 10, and 11. The y/D denotes the dimensionless distance along normal direction from -0.5 to 0.5 and the z/d denotes the dimensionless distance along the spanwise direction from -0.5 to 0.5. The DNS of Fang et al. [11] at the friction rotation number of 1 was chosen for comparison due to the lack of data of DNS of Fang et al. [11] at the friction rotation number of 1.3.

Figure 6. Mean secondary velocity vectors and dimensionless mean streamwise velocity contours at (a) $Ro_f = 0$; (c) $Ro_f = 1.3$; (e) $Ro_f = 5$, and Mean secondary streamline at (b) $Ro_f = 0$; (d) $Ro_f = 1.3$; (f) $Ro_f = 5$. 
3.2.1. Mean flow field
The mean flow field were obtained over a time period of 270 $D/u_\tau$ and 280 $D/u_\tau$ with the coarse mesh for the friction rotation numbers of 1.3 and 5, respectively. The mean velocity profiles along the wall bisectors in normal and spanwise directions of the present LES at $Ro_\tau = 0$, 1.3 and 5 were compared with DNS of Dai et al. [10] at $Ro_\tau = 1.3$ and 5 and DNS of Fang et al. [11] at $Ro_\tau = 1$ and 5. In addition, the friction velocity scales were used since the friction velocity can directly indicate the shear stress distribution and there were available DNS results of the flows.

![Figure 7. Comparison of wall shear stress of the present LES with DNS of Gavrilakis [2].](image)

Figure 7 shows the dimensionless mean streamwise velocity, $u^+$, profiles along the y axis at the wall bisector. The good agreement between the present LES and both DNS databases was obtained. For the present LES stationary duct case ($Ro_\tau = 0$), the profile is symmetrically distributed along the y axis. As the friction rotation number increases up to 1.3, the profile tend to move toward the pressure side on the bottom wall due to the Coriolis force. By further increases the friction rotation number up to 5, the magnitude of the profiles decreases while the profiles still move toward the pressure side.

Figures 9 shows the dimensionless mean streamwise velocity, $u^+$, profiles along the z axis at the wall bisector. The good agreement between the present LES and both DNS databases was obtained. The profile is symmetrically distributed along the z axis at $Ro_\tau = 0$. As the friction rotation number increases from 0 to 1.3, the profiles along the z axis at the wall bisector become flat with constant velocity in the central region of the duct. In addition, the constant velocity profile region along the spanwise direction represents the Taylor-Proudman region. By further increases the friction rotation number up to 5, the Taylor-Proudman region increasingly expands in the spanwise direction. However, the magnitude of the profiles decrease as the rotation number increases.

Figure 11 clarifies that the dimensionless bulk mean velocity, $U_b^+$ decreases as the rotation number increases. The good agreement between the present LES and DNS of Fang et al. [11] was obtained.

Figure 6 displays the mean secondary velocity vectors and the dimensionless mean streamwise velocity contours and the mean secondary velocity streamline. For the flows without rotation ($Ro_\tau = 0$), the turbulence induced secondary flows which consists of two counter rotating vortices distributed in each corner of the duct are presented. As the friction rotation number increases from 0 to 1.3, the vortices above the suction side are completely suppressed while the turbulence induced two counter rotating vortices on bottom wall of the stationary duct are reduced in size. Hence, the Coriolis force driven two large circulations with a pair of additional vortices on the pressure wall are presented. The Ekman layers are formed on the lateral walls. The descending current transfer the low momentum fluid from the suction side towards the pressure side while the ascending current appear on the lateral wall.
Figure 8. Turbulence structures with turbulence intensity contours (Q-criterion) at (a) $Ro_T = 0$; (b) $Ro_T = 1.3$; (c) $Ro_T = 5$. 
Figure 9. Comparison of mean vertical velocity profiles of the present LES with DNS of Dai et al. [10] and DNS of Fang et al. [11] at different friction rotation numbers.

Figure 10. Comparison of mean spanwise velocity profiles of the present LES with DNS of Dai et al. [10] and DNS of Fang et al. [11] at different friction rotation numbers.

The Taylor-Proudman region are found in the center region of the duct. By further increases the friction rotation number up to 5, the pair of additional vortices on the pressure wall are grown again. However, the mean streamwise velocity continuously decreases as the rotation number increases.

3.2.2. Turbulence structures

Figure 8(b) and figure 8(c) display turbulence structures with turbulence intensity contours for the flows inside the rotating duct at the friction rotation number of 1.3 and 5, respectively. The Q-criterion is used to expose the turbulence structures. As shown on both figure 8(b) and figure 8(c), the turbulence structures are totally suppressed on the top wall due to the effects of Coriolis force. Likewise, the turbulence intensity is weak on the top wall where the turbulent is suppressed whereas the high
turbulence intensity occurred on the bottom wall where the turbulent is enhanced. As the rotation number increases, the turbulence structures are much more stretched out on the top wall. By the way, the overall turbulence intensity is reduced as the rotation number increases due to the effects of Coriolis force. Therefore, the flows are laminarized at high rotation number.

4. Conclusion
In this paper, the incompressible fully developed turbulent flows in the stationary and rotating straight square ducts with the two different friction rotation number of 1.3 and 5 at the fixed friction Reynolds number of 300 was presented using ANSYS FLUENT 18.1’s LES with a dynamic kinetic energy subgrid scale model proposed by Kim and Menon [12]. The present LES results were compared with the available simulation databases of DNS and LES.

The good agreement between the present LES results and the reference data were obtained. However, the different result of the local maximum wall shear stress at the wall bisector of the stationary duct was obtained due to the sampling procedure.

For non-rotating duct ($Ro_\tau = 0$) case, the turbulence induced secondary flows are presented which consists of two counter rotating vortices distributed in each corner of the duct.

As the rotation number increases, the turbulence induced secondary vortices are suppressed on the suction side while reduced in size, at $Ro_\tau = 1.3$, and grown again, at $Ro_\tau = 5$, on the pressure side. Thereby, the Coriolis force driven two large circulations with the pair of additional vortices are presented. By the way, the Taylor-Proudman region is found in the center region of the duct and the Ekman layers are formed on the lateral wall. Furthermore, the turbulent flows are enhanced on the pressure side whereas the flows are laminarized on the suction side. Thus, the turbulence structures are stretched out on the suction side due to the effects of rotation. Moreover, the overall turbulence intensity is reduced as the rotation number increases. Hence, the flows are laminarized at high rotation number. However, the magnitudes of bulk streamwise velocity are found to be decreased as the rotation number increases due to the Coriolis effects.

Finally, the physics of the secondary flows in the stationary and rotating straight square ducts can be represented by using ANSYS FLUENT CFD software with the realistic accuracy of LES with one transport equation dynamic kinetic energy subgrid scale model.

![Figure 11. Comparison of dimensionless bulk mean velocity of the present LES with DNS of Fang et al. [11] at different friction rotation numbers.](image-url)
Acknowledgement
This work was supported by the graduate research assistant scholarship granted by the faculty of engineering, Kasetsart University. The simulations were performed on Wata Cluster supercomputer at the faculty of engineering, Kasetsart University.

References
[1] Madabhushi R K and Vanka S P 1991 Physics of Fluids A 3 pp 2734-2745
[2] Gavrilakis S 1992 J. of Fluid Mechanics 244 pp 101-129
[3] Huser A 1993 J. of Fluid Mechanics 257 pp 65-95
[4] Joung Y, Choi S U and Choi J I 2007 J. of Engineering Mechanics 133 pp 213-221
[5] Yao J, Zhao Y and Fairweather M 2015 Applied Thermal Engineering 91 pp 800-811
[6] Zhang H, Trias F X, Gorobets A, Tan Y and Oliva A 2015 Int. J. of Heat and Fluid Flow 54 pp 258-267
[7] Pallares J and Davidson L 2000 Physics of Fluids 12 pp 2878-2894
[8] Pallares J and Davidson L 2002 Physics of Fluids 14 pp 2804-2816
[9] Pallares J, Grau F X and Davidson L 2005 Physics of Fluids 17 pp 1-11
[10] Dai Y J, Huang W X, Xu C X and Cui G X 2015 Physics of Fl...