Accuracy of HiFiLES for DNS of compressible wall bounded turbulent flows

Nattakit Chaiwong*, Watchapon Rojanaratanangkule and Arpiruk Hokpunna

Department of Mechanical Engineering, Faculty of Engineering, Chiang Mai University
239 Huay Kaew Rd., Muang District, Chiang Mai, Thailand, 50200
* Corresponding Author: nattakit_ch@cmu.ac.th, +6686 6565 488

Abstract

In this work, we investigate a new HiFiLES code for Direct Numerical Simulation (DNS) of compressible wall-bounded turbulent flows. The HiFiLES-solver was developed from high-order numerical schemes based on an Energy-Stable Flux Reconstruction approach. This research is a verification of the HiFiLES performance for DNS study of turbulent compressible flows over wall bounded. This study selects a numerical method to find the approximation transformed discontinuous solution value of the Discontinuous Galerkin method (DG). The laminar test cases confirm a higher-order convergence of the code.

Keywords: HiFiLES, Energy-Stable Flux Reconstruction, Discontinuous Galerkin method, turbulent flow, compressible flow.

1. Introduction

Computational fluid dynamics (CFD) has played a major role in the equipment design in engineering since there are the obvious advantages when compared with an experiment method. The advantages are in various terms such as budget, location, equipment, and convenient of operation. Moreover, flow simulation can also study some type of flow that cannot be achieved in an experiment. However, flow simulation provides accurate results depending on various factors such as the numerical method used in the calculation, the performance of the calculating computer and complexity of the problem. The study of internal flow [8] is a major problem in engineering and this study aims to increase understanding about flow behavior which occurred nearby wall in order to enable equipment design and further development. Problems that usually use to study the flow through the wall are the flow through channel flow, the flow in circular pipe and the flow in the rectangular pipe. Although, these shapes are not very complicated for flow problems but still be the fundamental problems that attractive to researchers to study for developing knowledge and verifying the accuracy of new numerical method such as the direct numerical simulation of a compressible turbulent flow uses the Discontinuous Galerkin method: DG [9] for verifying the accuracy of the DG and studying the turbulent flow in a more realistic shape. Therefore, researchers have been developed numerical methods to calculate the accuracy of the results and introduced the numerical methods to create the realistic simulation and applied to various turbulent flow problems to enable researchers to understand the fluid behavior by studying the flow simulation.

In 2012 [1], the new calculation method for flow simulating called Energy Stable Flux Reconstructions (ESFR) approach was invented to solve flow problems by the fluid equation calculating which the Euler equation and Navior-stokes equation for 2D and 3D problems in order to verify the ability and accuracy. In 2013 [2], ESFR method for advection-diffusion problem was experimented. The result of this experiment has been guaranteed that the ability is stability and accuracy in every high order to find the solution and also be published to researchers for developing flows simulation. In 2014 [4], the ESFR calculation was developed to simplify for creating flow simulation and publishing. For the calculation of flow simulation can use the direct numerical simulation (DNS) and large eddy simulation (LES).The code name HiFiLES was created to be open-source software for researchers who interested in the development and deployment. This code is applied to experiment the accuracy of the simulation for the 2D and 3D flow.

The research of accuracy of HiFiLES for DNS of compressible wall bounded turbulent flows aimed to verify the solution accuracy and the code efficacy in the calculation. The study selects a numerical method to find the approximation transformed discontinuous solution value of the

Discontinuous Galerkin method (DG). The accuracy verification result from the laminar flow studied can be used to confirm the accuracy as the data to select the suitable high-order for the grid element in calculating, and can progress to further turbulent flow study.

2. Theory:

Fully develop flow in channel flow

The internal flow, generally the first flow can be classified into two regions. First region flow is entrance length that developing profile flow. During this length, the shape of the velocity will change at the time of the fluid moves deeper into the interior. When a fluid boundary layer thickness increases until squeeze and crash each other, then will approach to the second region flow which is fully develop flow and the shape of the fluid velocity in this second region is constant.

The relation of the fluid property flowing through the channel flow can be developed from the Navier-stokes equation that cannot use with incompressible fluid. It is a constant flow that flowing in the x-axis and can be written as below.

$$\rho u\frac{\partial u}{\partial x}+\rho v\frac{\partial u}{\partial y}=\rho g_x-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \quad (1)$$

From a conservation of mass equation can be written as

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \quad (2)$$

Fully develop flow boundary condition is the shape of the fluid velocity does not change when the fluid move along the x axis because the squeezing of the boundary layer make the velocity as zero on boundary and the new Eq.(2) which mean the value of velocity in the y direction is constant and not change by the y axis. Besides the condition of no-slip at wall makes the value of the fluid velocity in the y direction on flat wall equals zero and because of the velocity (v) is constant all over the cross-section of the channel between the plates. Thus, the velocity (v) in fluid is zero all over the cross-section. When consider Eq.(1) by using result of at the velocity of the y direction is zero and without considering the force from gravity force in the x-axis, then will be

$$\frac{\partial^2 u}{\partial y^2}=\frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right) \quad (3)$$

which is a differential equation form integrates the Eq.(3) to find the velocity u by the Eq.(4) two constant values are C1 and C2 will be given which can find from the set boundary condition.

$$u=\frac{1}{\mu}\left(\frac{\partial p}{\partial x}\right)\frac{y^2}{2}+C_1 y+C_2 \quad (4)$$

The channel flow in the y-axis = a and the x-axis is infinite value. The condition of channel flow boundary sets the top wall value as u = 0 at y = a and bottom wall value as u = 0 at y = 0.Then, we will find the shape of channel flow velocity as the Eq.(5).

$$u=\frac{a^2}{2\mu}\left(\frac{\partial p}{\partial x}\right)\left[\left(\frac{y}{a}\right)^2-\left(\frac{y}{a}\right)\right] \quad (5)$$

When use the Eq.(5) to calculate by the velocity profile condition, the shape will show as Fig.1.

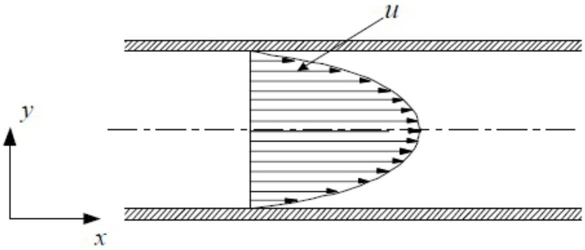

Fig.1 Velocity profile in channel flow [8]

The maximum velocity occurred in the middle of channel at $y=a/2$ and substituted in Eq.(4). Then,

$$u=u_{\max}=-\frac{1}{8\mu}\left(\frac{\partial p}{\partial x}\right)a^2=1.5\bar{V} \quad (6)$$

and the wall shear stress is

$$\tau=\mu\frac{du}{dy}=\frac{a^2}{2}\left(\frac{\partial p}{\partial x}\right)\left[\frac{2y}{a}-\frac{1}{a}\right] \quad (7)$$

The flow is a laminar flow. The result shows that the flow condition will change to turbulent flow when the Reynolds number is about 1,400.

Thus, for the analytical solution of the flow velocity through a channel flow in 2-D problem shows as

$$u(y)=\frac{3}{2}\bar{u}_{ave}\left(1-\frac{y^2}{h}\right) \quad (8)$$

3. Computational details

3.1 Numerical Methods: Energy Stable Flux Reconstruction approach

A study [1] ESFR schemes for advection-diffusion problem on HiFiLES code, shows the procedure of calculation in 1-D form. The calculation by Flux Reconstruction for 1-D problem starting from conservation law

$$\frac{\partial u}{\partial t}+\frac{\partial f}{\partial x}=0 \quad (9)$$

by means of x is the coordinate of X-axis , t is time, u is velocity which a function of $u(x,t)$ and

f is flux which a function of $f\left(u,\partial u/\partial x\right)$, considers under the domain (Ω) and set n as a position on a domain that does not overlap in each element at $\Omega_n=\left\{x\left|x_n<x<x_{n+1}\right.\right\}$.The solution of u under Ω_n is the approximate value from the functions that can be showed as $u_n^D=u_n^D\left(x,t\right)$ and be a polynomial degree P within Ω_n, by the superscript D indicates the discontinuity as u_n^D and u_{n+1}^D which the discontinuity occurs at the edge between the near element in a domain Ω_n and Ω_{n+1}. For the flux (f) approximate value in each domain Ω_n, from the function can be showed as $f_n^D=f_{n+1}^D\left(x,t\right)$. That is polynomial degree P + 1 within Ω_n and the superscript D value indicates the discontinuation as f_n^D and f_{n+1}^D have the discontinued at the edge between the near elements in the domain Ω_n and Ω_{n+1}.

Original form under Physical element Ω_n can transform coordinates into standard elements. The standard element value as $\Omega_s=\left\{r\left|-1\le r\le 1\right.\right\}$ by mapping in function of $\Theta_n\left(r\right)$

$$x=\Theta_n\left(r\right)=\left(\frac{1-r}{2}\right)x_n+\left(\frac{1+r}{2}\right)x_{n+1} \qquad (10)$$

applied Jacobian with the equation in mapping. Then,

$$\frac{\partial \hat{u}_n^D}{\partial t}+\frac{\partial \hat{f}}{\partial r}=0 \qquad (11)$$

Where $\hat{u}^D=\hat{u}^D\left(r,t\right)=J_n u_n^D\left(\Theta_n\left(r\right),t\right)$ and $\hat{f}=\hat{f}\left(r,t\right)=f_n\left(\Theta_n\left(r\right),t\right)$

Next, using method of the flux reconstruction the for polynomial and discrete solution value can be written in

$$\hat{u}^D=\sum_{i=0}^{P}\hat{u}_i^D l_i \qquad (12)$$

$$\hat{f}^D=\sum_{i=0}^{P}\hat{f}_i^D l_i \qquad (13)$$

The approximate solution at the edge of standard element Ω_s, the approximate transformed discontinuous solution at the edge of the Standard element has the left edge as $\hat{u}_L^D=\hat{u}^D\left(-1,t\right)$ and the right edge as $\hat{u}_R^D=\hat{u}^D\left(1,t\right)$ for calculation on the interface point, the study of HiFiLES code interests in 1-D LDG method. Solution of transformed continuous solution $\hat{u}=\hat{u}\left(r,t\right)$ which $\hat{u}$ has a polynomial degree equals P + 1 in the domain Ω_s that has the solution of the coordinate transformation from $\hat{u}_L^I$ and $\hat{u}_R^I$ in the left and right point respectively. The solution from the continuous coordinate transformation $\hat{u}$ is created from the polynomial degree equals P + 1 which from the calculation of $\hat{u}^c=\hat{u}^c\left(r,t\right)$ by $\hat{u}^c$ is the correction function that used to find the solution from transformed discontinuous solution $\hat{u}^D$ which include the summation equals the general solution of the coordinate transformation at element in left and right edges. The solution is under the domain Ω_s as following

$$\hat{u}^c=\left(\hat{u}_L^I-\hat{u}_L^D\right)g_L+\left(\hat{u}_R^I-\hat{u}_R^D\right)g_R \qquad (14)$$

By $g_L=g_L\left(r\right)$ and $g_R=g_R\left(r\right)$ are the correction function which polynomial degree P+1 approximate near zero under the domain Ω_s as

$$g_L\left(-1\right)=1, g_L\left(1\right)=0$$
$$g_R\left(-1\right)=0, g_R\left(1\right)=1$$

When consider with $\hat{u}=\hat{u}\left(r,t\right)$ in the domain Ω_s will have the transformed continuous solution and the transformed discontinuous solution include solution correction function as below

$$\hat{u}=\hat{u}^D+\left(\hat{u}_L^I-\hat{u}_L^D\right)g_L+\left(\hat{u}_R^I-\hat{u}_R^D\right)g_R \qquad (15)$$

The calculation of transformed numerical flux at the standard element domain Ω_s value of the solution is derived by approximating the transformed discontinuous solution value ($\hat{u}_i^D$) and transformed auxiliary variable. Calculation by transformed numerical interface flux by f_e^I will show calculation of the numerical flux at the interface e between the near elements of Ω_n and Ω_{n+1}. f_e^I will be calculated and made up from two parts.

1. An advective Part (Inviscid), $f_{e,\text{adv}}^I$
2. A diffusive Part (Viscous), $f_{e,\text{diff}}^I$

Thus, we will have an equation of $f_e^I=f_{e,\text{adv}}^I+f_{e,\text{diff}}^I$ value of $f_{e,\text{adv}}^I$ depending on

$u_{e,-}^{D}$ and $u_{e,+}^{D}$ for $f_{e,\mathrm{diff}}^{I}$ depending on $u_{e,-}^{D}, u_{e,+}^{D}, q_{e,-}^{D}$, $q_{e,+}^{D}$ for solving the linear advection-diffusion equation. In term value of advective interface flux $f_{e,\mathrm{adv}}^{I}$ will be calculated by using the Lax-Friedrich flux or using the Roe method, Rusanov method which the same as the Riemann-solver method. In term of the diffusive interface flux $f_{e,\mathrm{diff}}^{I}$ is discretized by DG. The numerical diffusive interface flux $f_{e,\mathrm{diff}}^{I}$ has format as

$$f_{e,\mathrm{diff}}^{I} = \left\{\left\{ f_{e,\mathrm{diff}}^{D} \right\}\right\} + \tau [\![u_e^D]\!] + \beta [\![f_{e,\mathrm{diff}}^{D}]\!] \quad (16)$$

The calculation of coordinates transformation at the interface flux in the standard element domain Ω_s shows as $\hat{f}_L^I$ and $\hat{f}_R^I$. For calculation has the Polynomial degree P + 1, the total flux will be the functions of $\hat{f} = \hat{f}(r,t)$ by the calculate to find $\hat{f}$ which consider at the P+ 1, the transformed correction flux will be $\hat{f}^c = \hat{f}^c(r,t)$. The approximate transformed discontinuous flux will be $\hat{f}^D = \hat{f}^D(r,t)$ and calculation that specified in $\hat{f}^c$ considers at P + 1, the value of correction function will be $h_L = h_L(r)$ and $h_R = h_R(r)$. When compared to the previous equation, it similar to calculation of g_L and g_R in form of approximation is zero in the domain.

$$h_L(-1) = 1, h_L(1) = 0$$

$$h_L(-1) = 0, h_L(1) = 1$$

For transform discontinuous flux value at the element right and left edge will be $\hat{f}_L^D = \hat{f}^D(-1,t)$ and $\hat{f}_R^D = \hat{f}^D(1,t)$ as

$$\hat{f}^c = \left(\hat{f}_L^I - \hat{f}_L^D\right) h_L + \left(\hat{f}_R^I - \hat{f}_R^D\right) h_R \quad (17)$$

The total continuous transformed flux value which a function of $\hat{f} = \hat{f}(r,t)$ in Ω_s can be calculated from combination of the discontinuous value and a correction flux value. Then, $\hat{f} = \hat{f}^D + \left(\hat{f}_L^I - \hat{f}_L^D\right) h_L + \left(\hat{f}_R^I - \hat{f}_R^D\right) h_R$ (18)

For the analysis about the discretization of Energy Stable Flux Reconstruction in this study, the calculation by HiFiLES code selects the correction function the Nodal Discontinuous Galerkin Scheme (DG). About the correction function, values of h_L and h_R are in the form of Flux correction function and has an important variable as C which a variable is a scalar function in the correction function.

$$\frac{-2}{(2P+1)\left(a_p P!\right)^2} < C < \infty \quad (19)$$

For the DG will set C equal to zero and the correction function in h_L and h_R will have an equation form as

$$h_L = \frac{(-1)^P}{2}\left(\psi_p + \psi_{p+1}\right) \quad (20)$$

$$h_R = \frac{1}{2}\left(\psi_p + \psi_{p+1}\right) \quad (21)$$

The polynomial degree which uses to calculate ψ_p is the Legendre polynomial of degree P.

The Energy Stable Flux Reconstructions method for the 2-D and 3-D uses the calculation same as the advection and diffusion in the 1-D form. Therefore, equation will be in the summation form and the calculation of code HiFiLES will use the ESFR methodology by discretize to find the solution of $\hat{u}^D$ $\hat{f}^D$ as

$$\hat{u}^D(\xi,\eta,\zeta) = \sum_{i=1}^{P+1}\sum_{i=1}^{P+1}\sum_{i=1}^{P+1} \hat{u}_{i,j,k}^{D} l_i(\xi) l_j(\eta) l_k(\zeta) \quad (22)$$

$$\hat{f}^D(\xi,\eta,\zeta) = \sum_{i=1}^{P+1}\sum_{i=1}^{P+1}\sum_{i=1}^{P+1} \hat{f}_{i,j,k}^{D} l_i(\xi) l_j(\eta) l_k(\zeta) \quad (23)$$

i, j, k are the index of the solution on the ξ, η, ζ direction. For calculating to find the boundary condition of correction function is the same. The calculation for 2-D and 3-D, h_i is the vector of correction function which depend on the edge of i (interface point i), $\vec{\xi}_j$ is position of the vector of j (interface point j), the n_j value is a vector 1 unit at the j point (interface point j) and δ_{ij} is the Kronecker delta for interface point will be a condition of calculation and can be written as

$$h_i\left(\vec{\xi}_j\right) \cdot \mathrm{n}_j = \delta_{ij} \quad (24)$$

3.2 Variables affecting the calculation by Energy Stable Flux Reconstruction approach

1. The location of solution point (r_i)
2. The methodology for calculating the transformed common solution values at $\hat{u}_L^I$ and $\hat{u}_R^I$ [CF, LDG, CDG, IP, BR1, BR2]
3. The methodology for calculating the transformed numerical interface fluxes at

$\hat{f}_L^I$ and $\hat{f}_R^I$ [a combination of Lax-Friedrich, Roe or Rusanov for advective part and CF, LDG, CDG, IP, BR1, BR2 for the diffusive part]

4. The form of solution correction function g_L and g_R
5. The form of solution correction function h_L and h_R

3.3 Physical and numerical parameters

In the study of accuracy verification of the HiFiLES code calculation which discretized from the ESFR and find the approximation transformed discontinuous solution value of the DG for verifying the accuracy of this problem, because of this flow problem has uncomplicated shape and calculation condition and the wall is two sides wall which has the top and the bottom and the fluid inlet and outlet in the left and right respectively. To verify the accuracy of the flow simulation at the Reynolds number set to 100 compares with bulk mean velocity. Due to this flow distributes and does not change by the Reynolds when it is a laminar flow, to calculate use the Navier - Stokes equation in 2-D and set the domain size calculation as 60 × 2 in the x and y axis and set the flow inlet is Mach number as 0.02. For the analytical solution of flow velocity through the channel flows in 2-D problem, the code computing set the grid elements number to be structure grid value are 60 × 2Y along the x and y axis, respectively and Y is variable value from 2, 4, 8, 16 and 32. The calculation verification at steady state and calculation using High-order is equal to 2, 3, 4, and 5.

4. Presentation and discussion result

The calculation of the code HiFiLES uses ESFR method for calculating the laminar flow problem by High-order and the grid element number. Change behavior that occurs at the inlet to the outlet of channel flow in the domain until be the steady state, this flow behavior is velocity changes from the incoming until changes when passed the incoming velocity. Calculation results are clearly consistent compared with the laminar flow theory when the occurring flow is fully-developed flow.

When use the velocity from domain of code HiFiLES in the y-axis to plot the velocity graph at the inlet and outlet in a steady state flow condition will find the velocity behavior changes at the inlet and outlet. Together with of the velocity development behavior at outlet as fully developed condition, and gives the results of velocity behavior which can be compared the theory solution as Fig.1.

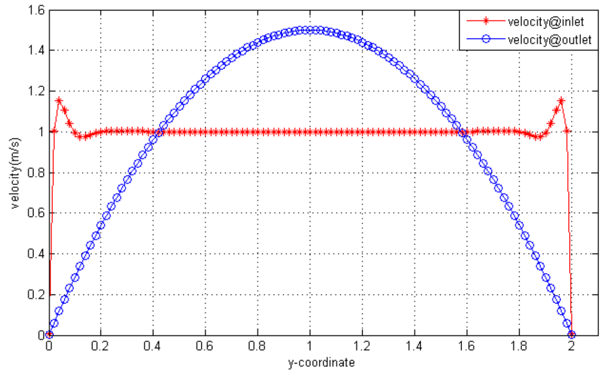

Fig.1 is a graph shows the velocity form in the y-axis about flow runs through the 2-D channel flow at the inlet (- * -, red) and outlet (-o-, blue).

For the velocity at the center of channel flow in each position on the x-axis at steady-state as Fig.2 which shows the changing behavior of the velocity from inlet range along the x-axis until converge to velocity about 1.5 m per sec. at x position equals 60, which is close to the analytical solution in the Eq.(8).

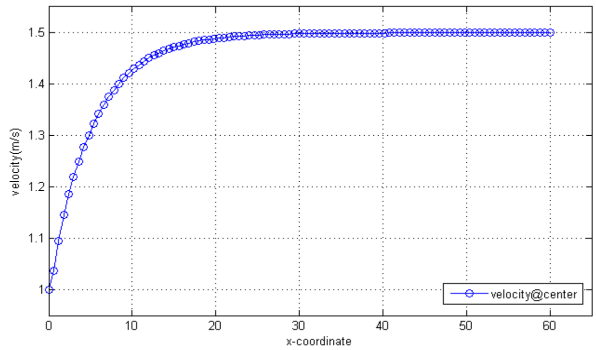

Fig.2 graph shows the change of the velocity in the x-axis which flow at the middle of 2-D channel flow.

When use all results from every cases of the code verification in order of accuracy of the study from high-order by plotting graphs between truncation error and grid size (dy), then plot by high order. Plot line graph to match with data and find the linear equation derived from a straight line. Graph results from calculating in each order will show as below.

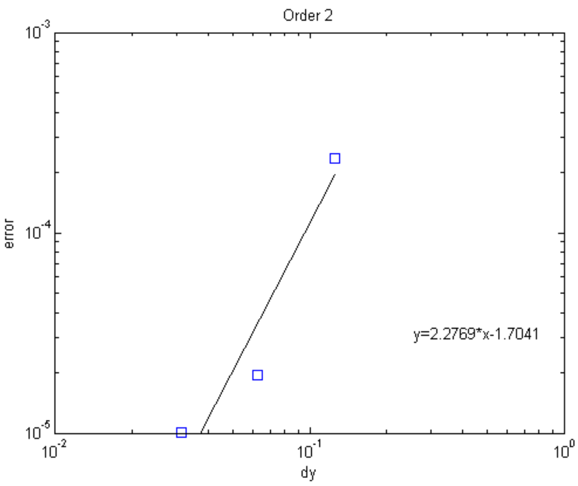

Fig3 (a) truncation error vs. grid size (dy) of Order 2

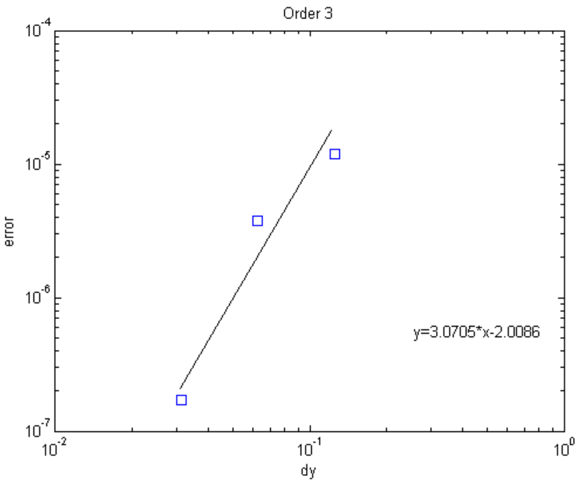

Fig3 (b) truncation error vs. grid size (dy) of Order 3

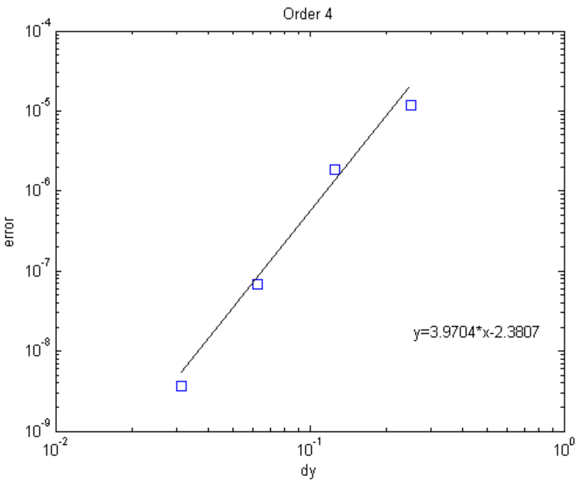

Fig3 (c) truncation error vs. grid size (dy) of Order 4

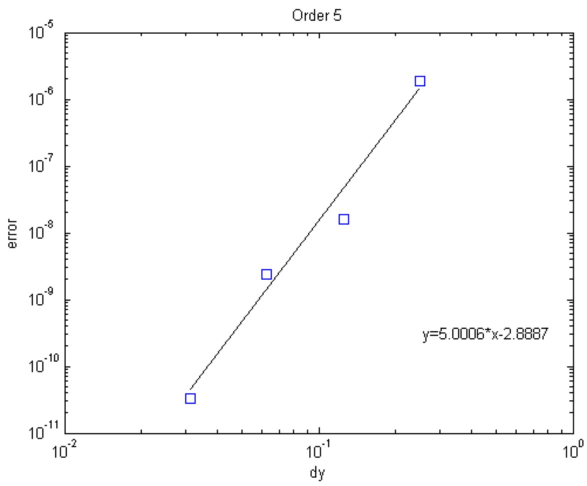

Fig3 (d) truncation error vs. grid size (dy) of Order 5

From the log-log graph between truncation error vs. grid size (dy) in each high-order. We will consider four different of order on study of performance of HiFiLES for DNS of compressible wall bounded turbulent flows and investigate their accuracy by varying the grid element (Y). The velocity value from calculation of HiFiLES code will be the approximate value when compared with the analytical solutions. The schemes under consideration are the second- , third-, fourth- and fifth-order. The exponent of grid element in y-axis is the order of accuracy of the method. It is useful measure of accuracy because it gives an indication of how rapidly the accuracy can be improved with refinement of the grid spacing. If we consider the mesh size by grid element (Y) is 4, 8, 16, 32 and 64, the truncation error is reduced to approximately follow quantity of grid element size. Since the errors are proportional to powers of grid size (dy), it is instructive to use a log-log plot to reveal the order accuracy of HiFiLES code. For each order, the curve representing the log error vs. the log dy is expected to be a straight line with its slope equal to the order of the high-order. The slope of the line straight in Figs.3 (a)-(d) verifies the order of each graph. The grid size value which from the width in the x-axis per the grid elements number in the x-axis equals 0.5, 0.125, 0.0625 and 0.03125.

However, when consider the CPU-time which calculated in each grid elements. It shows the calculation ability of HiFiLES code have the high accuracy and precision value, especially if use the high-order. But when compared to CPU-time, it requires time and resources more than using High-order to calculate.

5. Concluding remark

The validity verification of HiFiLES for DNS compressible wall bounded turbulent flows, which calculate the approximation transformed discontinuous solution value of the DG method for channel flow 2-D. This study provides the accurate result of flow simulation and shows the flow behavior in channel flow 2D which follow the flow theory. Moreover, this study increases the understanding of HiFiLES code using for the flow simulation and the effect on accuracy calculation of HiFiLES. The effects are result from using the grid element in the y-axis, result from the use of high-order to calculate, result from the complexity of the flow problems and result from the computer performance in computing.

HiFiLES in laminar flow problem can be a basis to validate and guarantee on research of DNS compressible wall bounded turbulent flows and can be used to select the suitable grid element number for the high-order in the flow simulation and to plan the resource management for using the computing computer and appropriate CPU-time.

6. Acknowledgement

Research on accuracy of HiFiLES for DNS of compressible wall bounded turbulent flows could not have been successfully accomplished without the supercomputer cluster support for computing the HiFiLES code accuracy was kindly provided by Department of Mechanical Engineering, Faculty of Engineering, Chiang Mai University.

7. References

[1] Castonguay P., *High-order energy stable flux reconstruction schemes for fluid flow simulations on unstructured grids*, Ph.D. thesis, 2012.

[2] Castonguay P., Williams D.M., Vincent P.E. and Jameson A., Energy stable flux reconstruction schemes for advection-diffusion problem, *Computer Methods in Applied on Mechanicals and Engineering*, Vol. 267, 2013, pp.400-417.

[3] Kim J., Moin P. and Moser R., Turbulence statistics in fully developed channel flow at low Reynold number, *Journal of Fluid Mech.*, Vol. 177, 1987, pp.133–166.

[4] Morales M. R. L., Bully J., Crabill J., Economon T. D., Manosalvas D., Romero J., Sheshadri A., Watkins II J. E., Williams D.M., Palacios F. and Jameson A., Verification and Validation of HiFiLES: a High-Order LES unstructured solver on multi-GPU platforms, *32nd-AIAA, Applied Aerodynamic Conference*, Atlanta, USA., AIAA paper 2014-3168.

[5] Moin P., *Fundamentals of Engineering Numerical Analysis*, 2nd edition, Cambridge university press, 2010.

[6] Ou K., Vincent P. and Jameson A., High-order method for diffusion equation with energy stable flux reconstruction schemes, 49th-*AIAA Aerospace Sciences Meeting including the New Horizons Exposit.*, Orlando, Florida, SA., January 4-7, 2011.

[7] Phamote Dechaumphai, *computational Fluid dynamics by Finite element and Finite volume Methods*, 2nd edition, Chulalongkorn University press, 2010.

[8] Somsak Chaiyapinunt, *Fluid Mechanic*, 3rd edition, Chulalongkorn university press, 2013.

[9] Wei L. and Polland A., Direct numerical simulation of compressible turbulent channel flows using the Discontinuous Galerkin method, *Journal of Computer and Fluid*, Vol. 47, 2011, pp.85-100.