HIGH-ORDER SIMULATION OF TURBOMACHINERY FLOW USING A DISCONTINUOUS GALERKIN METHOD

M. Yao
University of Oxford
Oxford, UK
(min.yao@eng.ox.ac.uk)

L. He
University of Oxford
Oxford, UK
(li.he@eng.ox.ac.uk)

ABSTRACT
A high-order unstructured implicit solver of the Reynolds-Averaged Navier-Stokes equations with the Spalart-Allmaras turbulence model has been developed based on the discontinuous Galerkin spatial discretization strategy. A range of validation studies have been performed. A first inviscid circular cylinder is simulated up to 5th order of accuracy and compared fairly well with the analytical solution. Another laminar flat plate boundary layer has also been validated by comparison with the standard Blasius solution. A further validation study of simulating the turbulent flat plate boundary layer has shown the capability of a high-order scheme in predicting the boundary layer with extreme coarse meshes, which alleviates the need of fine meshes commonly generated near the wall boundary. The Durham turbine cascade is finally tested by comparing the different orders of computational results with the experimental data. The high-order results have well resolved the flow field and captured the secondary flow vortices. The present results as a validation study have demonstrated the great potential capability of the high-order scheme in predicting the complex turbomachinery flow features and loss generation effectively with relatively coarse meshes.

INTRODUCTION
Turbo-machinery flow is inherently unsteady, multi-scale in both time and space with complex geometries, which tends to be more complicated and challenging than many other fundamental aerodynamic applications. The Computational Fluid Mechanics (CFD) has played a significant role in the design and analysis evaluation of performance of turbomachinery both in academic research and industrial Research and Design (R&D). These applications in the single- and multi-blade row designs, leakage flows, cavity flows and heat transfer have demanded more accurate predictive capability, which requires a high-resolution scheme due to the presence of some small-scale flow features. However, the conventional CFD codes as widely applied in industrial and academic settings are mostly based on 2nd order discretization schemes, which tend to be dissipative and often inadequate when pursuing high resolution of complex problems involving turbomachinery design and analysis. Hence, the need for developing and applying high-order schemes has been highlighted in pursuing high-resolution of practical turbomachinery problems.

Among these high-order schemes, the discontinuous Galerkin (DG) method has obtained the enormous interest due to its accuracy, robustness and flexibility to complex geometries. The high-order accuracy is achieved by high-order approximate piece-wise polynomials without depending on the stencils as in the conventional finite volume method (FVM). In addition, since the approximate solutions are represented by the piece-wise polynomials without inter-element continuity restrictions, the upwind flux schemes initially developed for FVM can be also adopted to compute the numerical flux through the element interfaces. The Runge-Kutta Discontinuous Galerkin (RKDG) is firstly developed to solve the system non-linear hyperbolic conservative laws in a general framework, particularly for the compressible Euler equations, with the explicit Total Variation Diminishing (TVD) Runge-Kutta method to achieve high-order temporal discretization (Cockburn et al., 1990). The RKDG evaluates the numerical flux using Riemann solvers in FVM essentially based on the framework of finite element method (FEM). The extension from the Euler to Navier-Stokes equations using DG discretization is pioneered by introducing the BR1 scheme of a mixed formulation to treat the viscous term for the spatial discretization (Bassi and Rebay, 1997). The RKDG for a purely hyperbolic systems is also extended to Local Discontinuous Galerkin (LDG) for convection-diffusion systems (Cockburn and Shu, 1998). The LDG handles the second-order viscous term by rewriting them into first-order equations and then employs the DG discretization for these first order systems. However, LDG introduces extended
stencils and increases the computational cost in the multidimensional applications. Thus, a new Compact Discontinuous Galerkin method (CDG) is developed for the elliptic problems, and the proposed schemes can remain compact stencils and reduce the sparsity pattern while retaining the advantages of LDG (Peraire and Persson, 2008). Also, the BR2 scheme is further developed which eliminates the extended stencils and only includes the nearest neighbour stencils (Bassi et al., 1997). Then, this BR2 scheme has been extended to discretize the Reynolds-Average Navier-Stokes (RANS) equations with a turbulence modelling closure. The DG discretization of compressible RANS equations with the $k-\omega$ turbulence model equations has been performed with an implicit time integration (Bassi and Rebay, 2005). Another approach to the numerical solutions of the RANS equations with the Spalart-Allmaras (SA) model uses the CDG scheme with additional shock-capturing artificial viscosity term (Nguyen et al., 2007; Oliver, 2008; Moro et al., 2011). Moreover, both BR2 and LDG are employed to discretize the RANS equations with the SA and the $k-\omega$ turbulence models (Landman, 2008).

For turbomachinery field, the development and applications of DG based solvers are relatively more recent. The direct numerical simulation (DNS) based on high-order DG has been simulated for a T106 low pressure turbine cascade (Garai et al., 2015; Garai et al., 2016). High-order DG methods have also been employed to simulate the turbulent flow through a MTU T106 low-pressure turbine cascade (Bassi et al., 2016). However, the high-order simulations have mostly been applied to predict the main flow features in low-pressure turbine flow fields and hardly predict the transitional suction boundary layer, and high-order schemes have also not been widely applied in secondary loss prediction and leakage flow simulations. Additionally, most of these high-order schemes have only been carried out for steady simulations.

The present development was initiated with an overall intent to develop an implicit unstructured mesh based DG solver for general turbomachinery applications. This paper describes the methodology and some validations studies for a turbomachinery configuration of practical interest.

METHODOLOGY

This section contains three parts, the discontinuous Galerkin discretization scheme, the time integration technique and the modification of a turbulence model for the DG method.

Discontinuous Galerkin Scheme

The RANS equations with the SA turbulence model are expressed in a conservative form as,

$$\frac{\partial U}{\partial t} + \nabla \cdot \left( F^c(U) - F^v(U,VU) \right) = S(U,VU).$$  \hspace{1cm} (1)

The computational domain $\Omega$ is partitioned into a set of non-overlapping elements $\Omega_k$ with boundary $\partial \Omega$. The approximate solution $U_h(t)$ in time space $t \in [0,T]$ belong to a finite three-dimensional space $V_h$.

$$\frac{\partial U}{\partial t} + \nabla \cdot \left( F^c(U) - V \cdot (\nabla U) \right) = S(U,VU).$$  \hspace{1cm} (2)

where $A$ is the Jacobian matrix of viscous flux. To obtain the approximate solution $U_h$, we multiply the governing equation (2) with test functions $v \in V_h$, integrate over the element and reach the DG formulation after doing integration by parts, as follows. Find $U_h \in V_h$, such that all $v \in V_h$,

$$\int_{\Omega} v^T \frac{\partial U_h}{\partial t} + R_{h,c}(U_h,v) + R_{h,v}(U_h,v) + R_{h,s}(U_h,v) = 0,$$  \hspace{1cm} (3)

where $R_{h,c}$, $R_{h,v}$ and $R_{h,s}$ represent the discretization of inviscid flux, viscous flux and source terms.

The inviscid flux discretization is given by:

$$R_{h,c}(U_h,v) = -\int_{\Omega} v^T \frac{\partial U}{\partial t} \cdot F^c(U_h) + \int_{\partial \Omega} v^T \frac{\partial F^c(U_h)}{\partial n} + \int_{\Omega} v^T \frac{\partial F^c(U_h)}{\partial n} - \int_{\partial \Omega} v^T \frac{\partial F^c(U_h)}{\partial n} \cdot n.$$

where $\partial \Omega$ denotes the interior faces, $\partial \Omega$ represents the boundary faces and $\vec{n}$ denotes the normal vector to the element boundary. We consider the Roe flux and Advection Upstream Splitting Method (AUSM) flux from approximate Riemann solver to evaluate inviscid fluxes here. In order to ensure the solver both works for an incompressible and a compressible flow, we employ a type of numerical fluxes based on the framework of AUSM scheme for all-speed flows (Liou, 2006). Next, the viscous flux is discretized by the second Bassi-Rebay method (BR2):

$$R_{h,v}(U_h,v) = -\int_{\Omega} v^T \frac{\partial U}{\partial t} \cdot (A \nabla U_h) - \int_{\partial \Omega} \left( (U_h^+ - U_h^-)^T \cdot \left( A \nabla U_h \right) + (v_h^T \cdot \left( (A \nabla U_h) - \eta_f(r_f) \right) \right) - \int_{\partial \Omega} \left( (U_h^+ - U_h^-)^T \cdot (A \nabla U_h^+) \cdot n^- + v_h^T \left( (F^+)^b - (F^-)^b \right) \cdot \eta_f r_f \right) \cdot n.$$  

where the $\{\} = 1/2(+) + (-)$ is an average operator and $[\cdot] = (-)n^- + (+)n^+$ denotes the jump operator. $r_f$ and $(F^+)^b$ are local lifting operator at interior and boundary faces respectively. The stabilization constant $\eta_f$ is set as 6 to meet the stability of BR2 discretization (Bassi et al., 1997).

Finally, the source terms are discretized as:

$$R_{h,s}(U_h,v) = -\int_{\partial \Omega} v^T \cdot S(U_h,VU_h).$$

Time Integration

The spatial discretization can lead to a system of ordinary differential equations (ODE) in time,

$$M \frac{dU}{dt} + R = 0.$$  \hspace{1cm} (4)
where $\mathbf{M}$ denotes the global mass matrix and $\mathbf{R}$ is the spatial residual vector. In order to eliminate the restrictions on the time step, we adopt an implicit integration technique for temporal discretization. Particularly, the backward Euler method is applied,

$$M \mathbf{U}^{n+1} - \mathbf{U}^n \Delta t + \mathbf{R} (\mathbf{U}^{n+1}) = 0. \quad (5)$$

Since the governing equations are non-linear, we linearize the residual vector $\mathbf{R}$ and obtain the linear algebraic system equations.

$$\mathbf{U}^{n+1} = \mathbf{U}^n - \left( \frac{1}{\Delta t} \mathbf{M} + \frac{\partial \mathbf{R}(\mathbf{U}^n)}{\partial \mathbf{U}^n} \right)^{-1} \mathbf{R}(\mathbf{U}^n). \quad (6)$$

They are solved using a preconditioned GMRES iterative algorithm with a block incomplete LU factorization with zero fill-in (ILU0) (Persson and Peraire, 2008).

**Modification of Turbulence Modelling**

The high-order methods solving the RANS equations normally meet with stiffness problems due to the non-smooth behaviour of turbulence modelling. In particular, for the SA turbulence model, the diffusion term $\nabla \cdot (\mathbf{v} \nabla \mathbf{v})$ in a laminar region is discontinuous in the first derivatives, which leads to the oscillations when applying a high-order discretization across the discontinuity.

Here, the eddy viscosity is computed as,

$$\mu_e = \rho \nu_v, \quad \nu_v = \bar{v} f_{v1}, \quad f_{v1} = \frac{\chi^3}{\chi^3 + C_{v1}}, \quad \chi = \frac{\bar{v}}{\nu_v}$$

The negative working variable $\bar{v}$ can result in the unbounded function $f_{v1}$ and failure of the computation. In addition, the negative $\bar{v}$ can cause negative total viscosity and turn dissipative terms into anti-dissipative terms. In order to prevent the negative $\bar{v}$, we employ the modification function to replace $\chi$:

$$\psi = 0.05 \log(1 + e^{20x}). \quad (7)$$

The proposed variable $\psi$ renders the results and deactivates the production, diffusion and destruction terms in the SA model when $\bar{v}$ becomes negative. Some recent work (Moro et al., 2011) has shown that this modified SA model has significantly enhanced the robustness of high-order RANS simulations.

**RESULTS AND DISCUSSION**

This section consists of four validation cases, an inviscid cylinder, a laminar flat plate boundary layer, a turbulent flat plate boundary layer and the Durham turbine cascade.

**Inviscid Flow around a Cylinder**

The first validation case is $M_e = 0.01$ flow around a circular cylinder. The geometry of cylinder is two-dimensional (2D) but the three-dimensional (3D) mesh is generated for our 3D solver computation by extruding a layer of mesh in third direction. Figure 1 shows the 3D computational mesh and there are only 16 elements to represent the cylinder. In order to refine the mesh at the boundary, high-order boundary representations are adopted and the curved boundary elements are represented by the same degree of polynomials as used in spatial discretisation (Landmann, 2008; Oliver, 2008). The simulations have been performed by different orders of approximate test functions from $p=0$ to $p=4$, where $p$ denotes the degree of polynomials with corresponding order of spatial discretization accuracy $p+1$. Figure 2 depicts the $p=1$ (2nd order) and $p=3$ (4th order) Mach number solution around the cylinder. It significantly illustrates that $p=3$ solution with the 4th order boundary representation is much better resolved than the $p=1$ solution with the 2nd order boundary.

**Laminar Boundary Layer over Flat Plate**

The second validation case is $M_e = 0.2, \text{Re} = 5 \times 10^3$ flow over a flat plate. As shown in Figure 4, the generated mesh (16 x 11 x 4, representing number of nodes in $x, y, z$ direction respectively) is relatively coarse. The first element spacing off the wall is $\Delta y/c = 4.56 \times 10^{-3}$ while the first element along the wall is $\Delta x/c = 3.85 \times 10^{-2}$. 

![Figure 1 Computational Mesh for Cylinder](image1)

![Figure 2 Computed Mach number contours](image2)
The similarity variable from the Blasius solution is 
\[ \eta = \frac{y}{(U_{\infty}/v)^{0.5}} \] 
and we plot it against the tangential velocity \( U_\tau / U_{\infty} \) and the non-dimensional normal velocity variable \( U_n / U_{\infty}(\text{Re})^{0.5} \). Figure 5 compares the computed tangential velocity and normal velocity profiles from \( p=2 \) and \( p=3 \) solution with the analytical ones from the Blasius at the 20% section of the plate. Clearly, the computed results from the high-order DGs match fairly well with the analytical one.

Figure 6 describes the comparison of the skin friction coefficient from \( p=1 \) and \( p=2 \) with the reference theoretical data (Schlichting, 1979). The computed skin friction coefficient from \( p=1 \) and \( p=2 \) have shown relatively reasonable agreement with the theoretical values. Figure 9 describes the comparison among \( p=1 \) and \( p=2 \) solution with the profile of the viscous sublayer and the log-law layer. The computational results match pretty well with the theoretical one.

In order to investigate the performance among different orders of spatial discretization, we now employ an extremely coarse node mesh (26x15x5) for the DG computations, as shown in Figure 10. The first element spacing off the wall is \( \Delta y / c = 1.12 \times 10^{-4} \) while the first element along the wall is \( \Delta x / c = 5.21 \times 10^{-4} \). Thus, the corresponding \( y^+ \approx 20 \) is much greater than the typical value required for a 2nd order fine volume discretization without invoking the wall
function. This coarse mesh however, can be adequately employed in a DG discretization, as shown below in Figure 10.

![Figure 10 Computed Mesh for Turbulent Flat Plate](image)

To illustrate the near wall behaviour as computed with different orders of accuracy, Figure 11 presents the non-dimensional velocity $u^+$ profile against $y^+$. The non-dimensional velocity from the second-order solution is markedly over-predicted in the logarithmic layer, while the solution get closer to the theoretical law of the wall as the discretization order increases. Overall, the fifth-order result predict the velocity profile quite well compared to the theoretical velocity in the viscous sub layer and the log-law layer. Thus, good results can be achieved by a high-order DG on very coarse meshes, whilst a second order scheme as widely adopted in conventional CFD methods clearly struggles with such coarse meshes, which will further alleviate the burden of generating quality meshes near solid walls of complex geometries for turbulent flow simulations.

![Figure 11 Computed Velocity Profiles](image)

### Durham Turbine Cascade

In this test case, the 3D turbulent flow past the Durham turbine cascade is simulated. The detailed geometry parameters are given in Table 1. The turbine configuration has a high flow turning angle, around 110°, which is similar to typical high-pressure axial flow turbine rotor blades. The secondary flow developed through this linear turbine cascade has been extensively investigated experimentally (Gregory-Smith et al., 1987). The experiments are conducted in a low-speed freestream condition with a flow velocity of 19.1 m/s. There are 11 traverse slots with hot wire and 5 hole pressure probe data in the flow passage to measure various flow variables, such as velocity, flow angles, pressure losses etc.

The numerical simulations of this linear cascade are conducted in a steady-state manner. In order to match the experimental boundary conditions, we gradually adjust the outlet static pressure to obtain the required inlet velocity. To achieve the measured inlet boundary layer profile, total pressure variation is specified as the inlet boundary condition. Also, the turbulent working variable $\nu'$ at inlet is calculated from the measured turbulence parameters (turbulence intensity and length scale) one axial chord upstream.
Accordingly, the inlet flow angle is slightly higher than the design value due to the deflection of upstream turbulence, which is set as 43.5° in the following DG simulations, in line with that recommended for CFD (Hartland et al., 1999).

Table 1 Geometrical design parameters of Durham turbine cascade

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet angle</td>
<td>42.75°</td>
</tr>
<tr>
<td>Outlet angle</td>
<td>68.7°</td>
</tr>
<tr>
<td>Axial Chord</td>
<td>181 mm</td>
</tr>
<tr>
<td>Pitch</td>
<td>191 mm</td>
</tr>
<tr>
<td>Half-Span</td>
<td>200 mm</td>
</tr>
<tr>
<td>Reynolds Number</td>
<td>$4 \times 10^5$</td>
</tr>
</tbody>
</table>

Figure 12 shows the computational mesh employed for the turbine cascade. This is a reasonably coarse mesh with a refined wake region downstream of the blade passage.

Figure 12 Computational mesh for Durham turbine cascade

The computations start with a low-order $p=0$ polynomial and increase to high-order simulations by adopting the restarted GMRES algorithm preconditioned by ILU0. Thus the results from the $p=0$ polynomial are regarded as the initialization for higher order simulations. The main flow patterns from different orders of the scheme can be observed in Figure 13. It depicts the contours of Mach number and total pressure ratio $p_t / p_0$ for $p=0$ to 2 polynomials. As the order of polynomial increases, the flow through the passage and the flow field around the leading edge, the trailing edge and wake region are notably much better resolved.

The computational results are mainly compared with the experimental ones at slot 10, located at the 28% axial chord downstream from the trailing edge of the cascade, since most detailed experimental data of the flow traverses are available there.

Figure 13 Computed Mach Number (up) and total pressure ratio (down) contours at mid-span from $p=0$ to $p=2$

Figure 14 presents the computed total pressure losses at the plane 28% axial chord downstream for different orders of polynomials. The measured experimental contours show that the passage vortex and the main counter vortex are located farther from the end wall, while a small counter vortex is observed in the crossflow region near the end wall. The computational results with the $p=1$ (2nd order accurate) polynomial have not captured the double loss peaks in the loss core. However, the results with the $p=2$ (3rd order accurate) polynomial distinctly predict the small counter vortex near the end wall and the double loss peaks of passage vortex, as well as the counter vortex further from the end wall. Therefore, it has clearly demonstrated that a high-order DG is capable of accurately capturing the vortices on a coarse mesh and can be applied to the prediction of secondary flow losses for turbomachinery flow.
In terms of the total pressure loss predictions however, Figure 15 indicates that the magnitude of the total pressure losses are still over-predicted by both the p=1 and 2 polynomials. This might be attributed to that the flow on the blade surface and end wall are mainly transitional while our present DG simulations adopting a fully turbulent flow (Moore and Gregory-Smith, 1996). As a result, the present DG simulations result in a larger total pressure loss coefficient.

Also, the pitch-averaged yaw angle distributions at the same section for both p=1 and p=2 show a reasonable match with the measured distribution.

The present validation study has shown some encouraging results for turbomachinery flow. Further validations and applications for steady and unsteady turbomachinery flows will be carried out in future work.

CONCLUSIONS

The RANS equations in conjunction with the Sparlart-Allmaras one equation turbulence model are discretized using a high-order Discontinuous Galerkin (DG) method on an unstructured mesh. The upwind schemes of AUSM and Roe have been applied for the inviscid flux calculations and the BR2 scheme has been implemented for the viscous flux discretization. An implicit time integration is adopted and the system linear equations are solved at each time step by using a preconditioned GMRES iterative algorithm with ILU0. A modification of turbulence modelling has been adopted to enhance the robustness for high-order turbulent flow simulations.

A range of validations studies have been carried out in the present work. Firstly an inviscid flow around a circular cylinder is validated, computational results from the high-order DG on a very coarse mesh have matched fairly well with the analytical solution, which demonstrates the proper implementation of convective fluxes. A laminar flat plate is then simulated, and the results are in agreement with the Blasius solutions, demonstrating that the viscous terms have been correctly discretized and implemented. A further turbulent flat plate boundary layer is presented with the modified SA model and the computational results from different orders of polynomials have shown good agreement.
with theoretical one. The results clearly reveal that the high-order DG is capable of predicting the boundary layer with a very coarse mesh, for which a conventional 2nd order scheme will not be adequate.

The final test case simulates a high turning turbine cascade and the present DG results are compared with the detailed experimental data, which shows an overall good match and indicates the capability of the high-order scheme in resolving the secondary flow field. Further validations and applications for a range of turbomachinery flows will be extensively carried out.

NOMENCLATURE
\( \bar{v} = \text{SA turbulence working variable} \)
\( v = \text{kinematic viscosity} \)
\( \mu = \text{eddy viscosity} \)
\( \mathbf{U} = \text{state variables vector} \)
\( \mathbf{F} = \text{inviscid flux} \)
\( \mathbf{F}^v = \text{viscous flux} \)
\( \mathbf{A} = \text{Jacobian matrix of viscous flux} \)
\( S = \text{source term} \)
\( \mathbf{U}_h = \text{approximate solution} \)
\( \mathbf{V}_h = \text{test function} \)
\( \mathbf{V}^3 = \text{finite three-dimensional space} \)
\( \partial \Gamma = \text{interior faces} \)
\( \partial \Omega = \text{boundary faces} \)
\( \mathbf{n} = \text{normal vector to element face} \)
\( \mathbf{r}^l = \text{local lifting operator} \)
\( \mathbf{M} = \text{global mass matrix} \)
\( R = \text{residual vector} \)

ACKNOWLEDGMENTS
The authors would like to express their appreciation for the support of China Scholarship Council, China and the support of the Computational Aerothermal Chair Studentship of Department of Engineering, University of Oxford.

REFERENCES