## Higher-Order Accurate Simulations of Blade-Vortex Interaction Using a Discontinuous Galerkin Method on Unstructured Meshes

Hee Dong Lee<sup>1</sup> Oh Joon Kwon<sup>2</sup>

Korea Advanced Institute of Science and Technology (KAIST) 373-1 Gusung-dong, Yusung-gu, Daejeon 305-701, KOREA

#### Abstract

A high-order accurate Euler flow solver based on a discontinuous Galerkin finite-element method has been developed for the numerical simulations of blade-vortex interaction phenomena on unstructured meshes. A free vortex in freestream was investigated to assess the vortex-preserving property and the accuracy of the present flow solver. Blade-vortex interaction problems in subsonic and transonic freestreams were simulated by adopting a dynamic solution-adaptive multi-level mesh refinement/coarsening technique. The results were compared with those of other numerical and experimental methods. It was shown that the present discontinuous Galerkin flow solver can preserve the vortex structure for significantly longer vortex convection time and can accurately capture the complex unsteady blade-vortex interaction flows, including generation and propagation of acoustic waves.

## 1. Introduction

In spite of the rapid development of modern CFD techniques, several difficulties still exist concerning accurate numerical simulations of unsteady flows involving convecting disturbances, such as vortices or acoustic waves, and their strong interactions with airfoil. One of the most prominent problems is the inherent numerical dissipation contained in CFD methods, which severely restricts preserving the strength of vortices, particularly at the far-field region away from the airfoil where grid resolution is not well established. To relieve this difficulty, special treatments, such as the perturbation method[1, 2] or the vorticity confinement method[3], have been used

<sup>2</sup> Corresponding author, Professor, Department of Aerospace Engineering, email: ojkwon@kaist.ac.kr. for solving blade-vortex interaction(BVI) problems. However, the perturbation method is not suitable for strong interaction cases, such as head-on vortexairfoil collision. The vorticity confinement method, which is similar to using an artificial viscosity for central-difference schemes, is not robust. It is better to reduce the numerical dissipation by increasing the accuracy of the numerical scheme and/or by improving the grid resolution in the local flow region where vortex convection and interaction is dominant.

On structured grids, several high-order spatially accurate methods were used to reduce the numerical dissipation for vortex flows [4-8]. Even though these high-order methods were successfully applied to better preserve the vortex structure than low-order accurate methods, enhancement of the local grid resolution in a dynamic manner is a very difficult task for structured grids. Using unstructured meshes, low dissipative solutions for vortex flows were obtained bv adopting solution-adaptive refinement techniques[9-11]. dvnamic mesh However, in the finite-volume frameworks, secondorder accurate methods are typically used for spatial discretization, and extension to high orders above second is virtually impractical on unstructured meshes, because excessive stencils are required for the reconstruction of high-order polynomials.

Recently, discontinuous Galerkin method(DGM) has experienced a resurgence of interest in various disciplines of numerical simulations on unstructured meshes[12, 13]. DGM has two advantageous features from both finite-element and finite-volume methods. In the case of DGM, high-order accuracy is achieved by increasing the degree of approximating polynomials without relying on extended stencils as in the classical finite-element methods. Since the approximate solutions are represented by elementwise polynomials, numerical flux schemes originally developed for finite-volume methods can be incorporated to determine unique flux values at the elemental boundaries. Also, DGM maintains the compactness, regardless of the order of accuracy,

<sup>&</sup>lt;sup>1</sup> Graduate Research Assistant, Department of Aerospace Engineering.

because the required stencil is confined only to the neighbours of elemental boundaries. Owing to this favourable property, DGM is highly parallelizable and easily handles adaptive mesh strategies. Therefore, high-order DGM coupled with a solutionadaptive dynamic mesh refinement technique is a good alternative to obtain low dissipative and highly accurate numerical solutions for vortex convection flows on unstructured meshes.

In the present study, a two-dimensional discontinuous Galerkin Euler flow solver has been developed on unstructured meshes for the accurate numerical simulation of vortex convection and interaction with airfoil. The nonlinear hyperbolic Euler equations in a differential form were recast to a form of а discontinuous weak Galerkin approximation. The approximate solutions were expressed by truncated polynomial series consisted of the hierarchical basis functions based on Legendre polynomials. Roe's approximate Riemann solver was applied to determine the interface flux. Time integration was achieved by using an explicit TVD Runge-Kutta method for unsteady timedependent problems, while an implicit time integration method based on Euler backward differencing was used to obtain steady-state solutions. A slope limiter[14] was adopted to suppress the non-physical oscillations of the flow in the vicinity of discontinuities, such as the shock wave. For the accurate capturing of the vortex structure and acoustic waves, a multi-level solutionadaptive mesh refinement technique based on a temporary cell approach was used[10, 11]. A freevortex convection problem in freestream[6] was investigated to assess the vortex-preserving property and the accuracy of the present high-order discontinuous Galerkin flow solver. Two-dimensional subsonic and transonic blade-vortex interactions were simulated, and the results were compared with other numerical simulations and experiment for validation.

## 2. Numerical Method

## 2.1 Governing equations

The two-dimensional Euler equations for compressible inviscid flows in a conservation form can be expressed as

$$\frac{\partial}{\partial t}\mathbf{U}(\mathbf{x},t) + \nabla \cdot \vec{\mathbf{F}} = 0, \quad (\mathbf{x},t) \in \Omega \times (0,T)$$
(1)

where  $\Omega \in \mathbb{R}^2$  is an open domain with boundary  $\partial \Omega \subset \overline{\Omega}$ . Also,  $t \in (0,T)$  represents time. The vector of conservative flow variables  $\mathbf{U}: \overline{\Omega} \times (0,T) \to \mathbb{R}^4$  and

inviscid Euler fluxes  $\vec{\mathbf{F}} = \{\mathbf{F}_1, \mathbf{F}_2\}; \mathbf{F}_j : R^4 \to R^4$  are defined as

$$\mathbf{U} = \begin{pmatrix} \rho \\ \rho u_i \\ \rho E \end{pmatrix}, \quad \mathbf{F}_j = \begin{pmatrix} \rho u_j \\ \rho u_i u_j + p \delta_{ij} \\ u_j (\rho E + p) \end{pmatrix}$$
(2)

where  $i = \{1, 2\}$ , and  $\rho$ , p and E denote density, pressure, and specific total energy, respectively.  $u_i$  is the velocity component in the Cartesian coordinate direction  $x_i$ , and  $\delta_{ij}$  represents the Kronecker delta. The governing flow equations are completed by the addition of the equation of state:

$$p = (\gamma - 1) \rho \left( E - \frac{1}{2} u_i u_i \right)$$
(3)

where  $\gamma$  is the ratio of specific heats.

#### 2.2 Discontinuous Galerkin formulation

Suppose that  $T_h$  is a subdivision of  $\Omega$  into disjunctive open element domains K such that  $\overline{\Omega} = \bigcup_{K \in T_h} \overline{K}$ . Also, each  $K \in T_h$  is an image of a reference element  $\hat{K}$ ,  $K(\mathbf{x}) = F_K(\hat{K}(\xi))$ , for all  $K \in T_h$ . The mapping  $F_K$  of the reference element  $\hat{K}(\xi)$  to the element  $K(\mathbf{x})$  in real space is assumed to be bijective and smooth. On the reference element  $\hat{K}$ ,  $P^p(\hat{K})$  is defined as the space of polynomials of degree  $p \ge 0$  with polynomial basis functions  $\hat{\phi}_j$ , and  $P^p(K)$  is the space of functions associated with the space  $P^p(\hat{K})$  through the mapping  $F_K$ :

$$P^{p}\left(\hat{K}\right) = span\left\{\hat{\phi}_{l}, l = 1, \dots, N_{dof}\right\}$$

$$P^{p}\left(K\right) = span\left\{\phi_{l} = \hat{\phi}_{l} \circ F_{K}^{-1}, l = 1, \dots, N_{dof}\right\}$$
(4)

The finite element space  $V_h^p$  consisted of discontinuous vector-valued polynomial functions of degree  $p \ge 0$  is defined as

$$V_{h}^{p} = \left\{ \mathbf{v} \in \left[ L_{2} \left( \Omega \right) \right]^{4} : \mathbf{v} \Big|_{K} \in \left( p_{1}, \dots, p_{4} \right)^{T} \right\}$$
$$p_{i} \in P^{p} \left( K \right)$$
(5)

Then  $\mathbf{U}(\mathbf{x},t)$  can be approximated by  $\mathbf{U}_{h}(\mathbf{x},t) \in V_{h}^{p}(K) \otimes C^{1}[0,T]$  as

$$\mathbf{U}_{h}\left(\mathbf{x},t\right) = \sum_{l=1}^{N_{dof}} \hat{\mathbf{U}}_{l}\left(K,t\right)\phi_{l}\left(\mathbf{x}\right)$$
(6)

where  $N_{dof}$  is the number of degree of freedoms to represent the approximate solution in a form of truncated polynomial expansions and is given by (k+1)(k+2)/2 for two-dimensional elements. The present DGM is different from standard continuous Galerkin methods in that expansion of  $\mathbf{U}_{h}$  is local for each element, without any continuity constraint across the element boundaries.

To obtain a weak form of the governing equations, the conservation laws in Eq. (1) are multiplied by an arbitrary smooth function **W** and are integrated by parts over an element *K* in the mesh  $T_k$ :

$$\frac{\partial}{\partial t} \int_{K} \mathbf{W}^{T} \mathbf{U} d\Omega + \int_{\partial K} \mathbf{W}^{T} \left( \mathbf{n}^{T} \vec{\mathbf{F}} \right) dS - \int_{K} \nabla \mathbf{W}^{T} \cdot \vec{\mathbf{F}} d\Omega = 0$$
(7)

where n denotes the outward unit normal vector to  $\partial K$  .

To discretize Eq. (1), an analytical solution U and the arbitrary function W are replaced by Galerkin finite-element approximation  $U_h$  and  $W_h$ , respectively, which belong to the finite-element space  $V_h^k$ . In addition, since the numerical solution  $U_h$  is discontinuous across the element interfaces, the flux  $\mathbf{n}^T \vec{\mathbf{F}}$  must be replaced by a numerical flux  $\mathbf{H}(\mathbf{U}_h^-, \mathbf{U}_h^+, \mathbf{n})$ , which depends on both inner and outer traces of  $U_h$  on  $\partial K$  and the unit outward normal vector  $\mathbf{n}$ . In the present study, the numerical flux of Roe[15] originally developed for finite-volume methods is used. The discontinuous Galerkin formulation of the governing equations can be expressed as follows:

$$\frac{\partial}{\partial t} \int_{K} \mathbf{W}_{h}^{T} \mathbf{U}_{h} d\Omega + \int_{\partial K} \mathbf{W}_{h}^{T} \mathbf{H} \left( \mathbf{U}_{h}^{-}, \mathbf{U}_{h}^{+}, \mathbf{n} \right) dS - \int_{K} \nabla \mathbf{W}_{h}^{T} \cdot \vec{\mathbf{F}} d\Omega = 0, \ \forall \mathbf{W}_{h} \in V_{h}^{P}$$
(8)

By introducing the polynomial expansions for  $U_h$  and  $W_h$  into Eq. (8), a set of equations for the coefficients  $\hat{U}_{il}$  is obtained:

$$\int_{K} \phi_{l} \phi_{m} d\Omega \,\delta_{ij} \,\frac{d}{dt} \hat{U}_{jm} = -\int_{\partial K} \phi_{l} H_{i} \left( \mathbf{U}_{h}^{-}, \mathbf{U}_{h}^{+}, \mathbf{n} \right) dS + \int_{K} \nabla \phi_{l} \cdot \vec{F}_{i} d\Omega$$
(9)

where  $i, j = \{1, ..., 4\}$  represent indices of the four Euler equations in two dimensions, and  $l, m = \{1, ..., N_{dof}\}$  denote indices of the basis functions and the coefficients of the truncated polynomial expansions. Equation (9) can be rewritten for an arbitrary element *K* as follows:

$$m_{lm}\delta_{ij}\frac{d\hat{U}_{jm}}{dt} = R_{il}\left(\mathbf{U}_{h}^{-},\mathbf{U}_{h}^{+}\right)$$
(10)

where

$$m_{lm} = \int_{K} \phi_{l}^{-} \phi_{m}^{-} d\Omega$$

$$R_{il} = -\int_{\partial K} \phi_{l} H_{i} \left( \mathbf{U}_{h}^{-}, \mathbf{U}_{h}^{+}, \mathbf{n} \right) dS + \int_{K} \nabla \phi_{l} \cdot \vec{F}_{i} d\Omega$$
(11)

Here, - and + denote the quantities inside and outside of K.

The relationship given in Eq. (10) for all elements in  $T_h$  can be expressed as a system of ordinary differential equations:

$$\mathbf{M}\frac{d\hat{\mathbf{U}}}{dt} = \mathbf{R}\left(\hat{\mathbf{U}}\right) \tag{12}$$

where M denotes the block-diagonal mass matrix,  $\hat{U}$  is the global vector of the coefficients, and R represents the residual vector.

This scheme is called as a discontinuous Galerkin method of degree p, or 'DG(p) method', which has an order of accuracy of p+1 [16]. This discontinuous Galerkin discretization is similar to finite-volume methods(FVM), especially in the use of the numerical fluxes, and in fact the first-order Godunov-type FVM is exactly identical to DG(0) method based on piecewise constants. Consequently, DG(p) method with p > 0 can be regarded as the natural generalization of FVM to high orders.

#### 2.3 Time integration

Once spatial discretization is completed, a set of ordinary differential equations is obtained as in Eq. (12). These ordinary differential equations can be integrated in time by using either an explicit or implicit method. Explicit time integration methods are more attractive for time-dependent unsteady flow problems, since they can easily be implemented to achieve high-order temporal accuracy. In the present study, a third-order accurate TVD Runge-Kutta method [17] was used as follows:

$$\hat{\mathbf{U}}^{(1)} = \hat{\mathbf{U}}^{n} + \Delta t \ \mathbf{M}^{-1} \ \mathbf{R} \Big( \hat{\mathbf{U}}^{n} \Big)$$

$$\hat{\mathbf{U}}^{(2)} = \frac{3}{4} \hat{\mathbf{U}}^{n} + \frac{1}{4} \hat{\mathbf{U}}^{(1)} + \frac{1}{4} \Delta t \ \mathbf{M}^{-1} \ \mathbf{R} \Big( \hat{\mathbf{U}}^{(1)} \Big)$$

$$\hat{\mathbf{U}}^{n+1} = \frac{1}{3} \hat{\mathbf{U}}^{n} + \frac{2}{3} \hat{\mathbf{U}}^{(2)} + \frac{2}{3} \Delta t \ \mathbf{M}^{-1} \ \mathbf{R} \Big( \hat{\mathbf{U}}^{(2)} \Big)$$
(13)

As the order of the polynomial basis increases, stability of the explicit time-marching scheme degrades rapidly[12]. As a result, the convergence may become dramatically slow for large-scale flow simulations. In the case of steady-state flow problems, a fully implicit method based on the backward Euler time integration was applied to Eq. (10) as

$$m_{lm}\delta_{ij}\frac{\hat{U}_{jm}^{-}\Big|^{n+1}-\hat{U}_{jm}^{-}\Big|^{n}}{\Delta t}=R_{il}^{n+1}\left(\mathbf{U}_{h}^{-},\mathbf{U}_{h}^{+}\right)$$
(14)

The right-hand side is now linearized by using the Taylor expansion as

$$R_{il}^{n+1}\left(\mathbf{U}_{h}^{-},\mathbf{U}_{h}^{+}\right) \approx R_{il}^{n} + \frac{\partial R_{il}}{\partial \hat{U}_{jm}^{-}} \bigg|^{n} \Delta \hat{U}_{jm}^{-} + \frac{\partial R_{il}}{\partial \hat{U}_{jm}^{+}}\bigg|^{n} \Delta \hat{U}_{jm}^{+}$$
(15)

where  $\Delta \hat{U}$  indicates  $\hat{U}^{n+1} - \hat{U}^n$ . The Jacobians of  $R_{il}$  to  $\hat{U}$  can be derived from Eq. (10) as

$$\frac{\partial R_{il}}{\partial \hat{U}_{jm}^{-}} = -\int_{\partial K} \phi_{l}^{-} \phi_{m}^{-} \frac{\partial H_{i}}{\partial U_{j}^{-}} dS + \int_{K} \phi_{m}^{-} \left( \frac{\partial \phi_{l}^{-}}{\partial x_{1}} \frac{\partial F_{1i}}{\partial U_{j}^{-}} + \frac{\partial \phi_{l}^{-}}{\partial x_{2}} \frac{\partial F_{2i}}{\partial U_{j}^{-}} \right) d\Omega \qquad (16)$$
$$\frac{\partial R_{il}}{\partial \hat{U}_{jm}^{+}} = -\int_{\partial K} \phi_{l}^{-} \phi_{m}^{+} \frac{\partial H_{i}}{\partial U_{j}^{+}} dS$$

where  $\partial H_i / \partial U_j$  is the Jacobian of Roe's numerical flux. Also,  $F_{1i} / U_j$  and  $F_{2i} / U_j$  are the Jacobians of the inviscid Euler flux in  $x_1$  and  $x_2$  directions, respectively. Then, Eq. (14) becomes

$$\left(m_{lm}\frac{\delta_{ij}}{\Delta t}-\frac{\partial R_{il}}{\partial \hat{U}_{jm}^{-}}\right)^{n}\Delta \hat{U}_{jm}^{-}-\frac{\partial R_{il}}{\partial \hat{U}_{jm}^{+}}\right)^{n}\Delta \hat{U}_{jm}^{+}=R_{il}^{n} \quad (17)$$

which can be rewritten in a matrix form for all  $K \in T_h$  as

$$\mathbf{A}\Delta \mathbf{U} = \mathbf{R} \tag{18}$$

The coefficient matrix **A** is a very sparse block matrix which is consisted of one diagonal block and three off-diagonal blocks for a triangular element. Each block is a square matrix having  $4N_{dof} \times 4N_{dof}$  entities. The linear system of equations is solved at each time step by using a point Gauss-Seidel method.

# 2.4 Basis functions, mappings, and quadrature rules

For the unstructured mesh used in the present study, each triangular element *K* in the real space of  $(x_1, x_2)$  can be considered as an image of the reference element in space  $(\xi_1, \xi_2)$ , as shown in Fig. 1. The domain of the reference element  $\hat{K}$  is defined as

$$\hat{K} = \left\{ \boldsymbol{\xi} \in R^2; -1 \le \xi_1, \xi_2; \xi_1 + \xi_2 \le 0 \right\}$$
(19)

The reference element is equipped with affine coordinates[18]

$$\lambda_1 = \frac{1+\xi_2}{2}, \lambda_2 = -\frac{\xi_1+\xi_2}{2}, \lambda_3 = \frac{1+\xi_1}{2}$$
 (20)

The approximate solution is expressed in a form of truncated polynomial expansions as in Eq. (6). The choice of the basis functions does not affect the accuracy of the numerical scheme, but a suitable choice of them may simplify the implementation and the calculation. In the present study, hierarchical basis functions based on the Legendre polynomials[19], which are adequate for local prefinement method, were used:

$$\phi_{l} = L_{n_{1}} \left( \lambda_{3} - \lambda_{2} \right) L_{n_{2}} \left( \lambda_{2} - \lambda_{1} \right), 0 \le n_{1}, n_{2}; n_{1} + n_{2} \le p$$
(21)

where L(x) denotes the Legendre polynomial, and  $\lambda_i$  is an affine coordinate defined in Eq. (20). Figure 2 shows a set of basis functions for DG(3) method, where the number of basis functions  $N_{dof}$  is equal to 10.

Since the basis set is defined on the reference element  $\hat{K}$ , the mapping  $F_{\kappa}$  is required to compute the derivatives and the integrals in the physical element K. For straight-sided elements, the mapping is linear and can be simply expressed by using the affine coordinates:

$$\mathbf{x} = F_K \left( \boldsymbol{\xi} \right) = \lambda_2 \mathbf{x}_1 + \lambda_3 \mathbf{x}_2 + \lambda_1 \mathbf{x}_3$$
(22)

where the Jacobian and the metrics of linear mapping are constants within each element. In the presence of curved boundaries, however, a meaningful high-order accurate solution can be obtained only if a corresponding high-order approximation of the curved-sided elements is employed[19]. In the present study, transfinite interpolation schemes with the aid of collocation projection[18] were applied to obtain the high-order mapping  $F_{\kappa}$  for curved-sided elements.

The set of Eq. (10) is solved in the reference element by the mapping  $F_{\kappa}$ , and the integrals are numerically evaluated by using the Gaussian quadrature rules. This requires projection of the solution values to the quadrature points used in the numerical integration. It is known that the quadrature rules for the surface integral in Eq. (11) must be exact for polynomials of degree 2p+1, and the quadrature rules for volume integral in Eq. (11) must be exact for polynomials of degree 2p, if DG(p) method is used[12]. In the present study, Gaussian quadrature rules provided in Ref. [20] were used.

#### 2.5 Slope limiter and shock detector

One of the main interests of compressible flow simulations is to accurately resolve discontinuities, such as the shock wave, which exist inside the flow field. It is well known that high-order schemes produce spurious oscillations in the vicinity of the discontinuities. To prevent non-physical oscillation near the discontinuities and to stabilize the numerical solution, either slope limiters[12, 20] or artificial viscosities[21-23] can be commonly applied. Both approaches have pros and cons. Even though slope limiters have been more widely used in DGMs, they may result in deterioration of the solution accuracy, even in smooth flow regions by clipping smooth extrema. Moreover, slope limiters prevent solution convergence to steady state, since the limiters are not integrated in the computation of the residual but used as a post-processing filter [22]. Therefore, application of slope limiters has been limited to unsteady flow problems when an explicit Runge-Kutta time integration method is used. The artificial viscosity approach was originally designed to stabilize high-order finite-element methods, such as the streamline upwind Petrov Galerkin(SUPG) method, by adding an artificial viscosity term into the weak formulation explicitly. This approach is very attractive when steady-state solutions are sought, but requires adjusting user-defined parameters which depend on both mesh and flow condition.

In the present study, a slope limiter[14] which was primarily developed for finite-volume methods was adopted to suppress spurious oscillations. To prevent the accuracy loss by clipping smooth extrema, the limiter was set to be active only in the vicinity of discontinuities as they are detected by a simple criterion based on the density jump across the elemental boundaries as follows:

$$\sigma_{K} = \sum_{e=1}^{n_{e}} \frac{1}{\left|\partial K_{e}\right|} \int_{\partial K_{e}} \frac{\left|\rho_{h}^{-} - \rho_{h}^{+}\right|}{\left|\rho_{avg}\right|} dS$$
(23)

where  $\partial K_e$  and  $n_e$  are segment of elemental boundaries and the number of segments, respectively.  $\rho_h^-$  and  $\rho_h^+$  represent the inside and outside values of density of an element K on  $\partial K_e$ , and their arithmetic average is denoted as  $\rho_{avg}$ . The limiter and the shock detector used in the present study were not optimized. Development of robust shock detectors as well as stabilization methods remains one of the challenging issues for obtaining high-order accurate solutions of DGM [24-26].

#### 2.6 Dynamic mesh adaptation

To improve the accuracy of flow simulation and to reduce the inherent numerical dissipation, a solutionadaptive mesh refinement/coarsening technique was applied[8]. For this purpose, grid points were dynamically added and deleted on the existing unstructured mesh as detected by error indicators. In the present DGM, difference of the approximate solution at the elemental boundaries was taken at the error indicator. This difference decreases as the discretization error is reduced when the spatial accuracy is increased or when the mesh resolution is enhanced. Therefore, magnitude of the difference can be considered as a suitable measure of the local discretization error. In the present study, a simple error indicator is adopted based on the difference of density along elemental boundaries as follows:

$$\varepsilon_{K} = \sum_{e=1}^{n_{e}} \int_{\partial K_{e}} \left| \rho_{h}^{-} - \rho_{h}^{+} \right| dS$$
(24)

Since the starting vortex is initiated at the far upstream from the airfoil in an analytic form, little solution difference exists across the elemental boundaries. To enhance the grid resolution at this initial stage, vorticity magnitude was also used as an error indicator, in addition to density difference.

Dynamic mesh adaptation is initiated on a relatively coarse mesh, and the cell division continues until a given criterion is satisfied. The error indicator is computed at each element, and then is transferred to the elemental face to decide whether cell division is necessary for each particular face. Then, new node points are added at the centers of tagged faces. To increase the grid efficiency and to reduce the computational cost, a mesh coarsening procedure is also applied in parallel to the mesh enrichment whenever the value of error indicator decreases due to the changing flow characteristics in time. During the mesh coarsening procedure, node points previously added are deleted and subdivided elements are restored to their original elements. The mesh enrichment history stored in the tree data structure is used for this purpose. Mesh coarsening is stopped when the original mesh is recovered.

#### 2.7 Vortex model

For the vortex convection problems, a vortex was initially described by an analytical form as suggested by Sculley[27]. The non-dimensional tangential velocity of the vortex is expressed as

$$v_{\theta} = \frac{\Gamma}{2\pi r} \left( \frac{r^2}{r^2 + r_c^2} \right)$$
(25)

where  $r_c$  and r represent the radius of vortex core and the radial distance from the vortex center, respectively. The vortex strength  $\Gamma$  is normalized by the product of freestream Mach number, freestream speed of sound, and airfoil chord length. Counterclockwise vortex is defined as positive in the present study. The initial pressure and density fields can be evaluated from the radial momentum equation, in conjunction with the energy equation for constant enthalpy flows[28].

## 3. Results and discussion

To assess the solution accuracy of the present high-order discontinuous Galerkin flow solver, unsteady time-accurate flows involving vortex convection were calculated. Initially, a free vortex in freestream was investigated to assess the vortexpreserving property of the present flow solver. Then, blade-vortex interaction problems in subsonic and transonic freestreams were simulated by adopting a multi-level solution-adaptive dynamic mesh refinement/coarsening technique.

## 3.1 Vortex convection in freestream

The first validation of the present method was made for a free vortex convecting in freestream. This case was first attempted by Tang and Baeder[6] to measure the diffusive property of spatial discretization methods on structured grids. The vortex model presented in Eq. (25) with a core radius of  $r_{a} = 0.05$  and a non-dimensional strength of  $\Gamma = -0.2$  was initialized at incoming upstream of a freestream Mach number of 0.5, and then the vortex was convected over a non-dimensional distance of 10, which is equivalent to 200 vortex core radii. The normalized size of the computational domain was  $20\!\times\!2$  , and the domain was tessellated into a uniform Cartesian mesh consisted of 241×41 points. To construct a triangular unstructured mesh, each quadrilateral cell was further divided into two triangles as shown in Fig. 3. As shown in the figure, the mesh was very coarse with a typical edge size equivalent to the radius of the vortex core. Unsteady time-accurate calculations were performed by using a third-order accurate TVD Runge-Kutta time integration method.

In Fig. 4, decay of the peak-to-peak tangential velocity and the minimum density difference normalized by the initial values are presented in terms of the convection distance. Figure 5 shows the tangential velocity and density profiles after the vortex convected 200 core radii(x=10). The figures show that numerical diffusion of the vortex was drastically decreased as the order of accuracy increased. In the case of the fifth-order calculation denoted as DG(4), the vortex was convected without any visible difference in the peak-to-peak tangential velocity from the exact value, even after 200 core radii travel. Also, only three percent decay was observed in the minimum density difference. The vortex core radii measured by the distance from the

position of the maximum tangential velocity to the vortex center were 1.0, 1.25, 1.67, 3.0 radii of the initial vortex for DG(1), DG(2), DG(3) and DG(4), respectively. This indicates that high-order accurate DGMs preserve the vortex without any significant decay, even on very coarse meshes.

Assessment of the accuracy of the present DG solver are presented in Table 1 and the results are compared with those of an improved third-order accurate upwind spatial discretization method developed by Tang and Baeder[29] based on a piecewise quadratic reconstruction with more accurate slopes and curvatures evaluated by compact difference schemes. The results compared were calculated on a uniform Cartesian mesh containing 481×81 points, four times more grid points than the present unstructured mesh. It is shown that the results based on the improved quadratic reconstruction method with sixth- and fourth-order compact difference schemes (Q6c and Q4c, respectively) are more accurate than other high-order methods, such as the fifth-order accurate WENO5[30] and the third-order accurate MUSCL scheme implemented in CFL3D[31], although the nominal accuracy is equivalent or lower. However, the results clearly indicate that the present DG(3) and DG(4) methods are less diffusive than both Q4c and Q6c, even on a relatively coarse mesh. The solution of the second-order accurate DG(1) method is still more accurate than that of CFL3D, even at lower order of accuracy.

## 3.2 Subsonic blade-vortex interaction

Next, a subsonic BVI problem was simulated by using the present high-order DG solver coupled with a multi-level solution-adaptive dynamic mesh refinement/coarsening technique. This quasi twodimensional head-on parallel blade-vortex previously interaction was investigated experimentally and numerically [28]. In this problem, a vortex with a core radius of 0.018 chord lengths and a non-dimensional strength of -0.283 was released upstream of an NACA0012 airfoil at a freestream Mach number of 0.5. An isotropic triangular mesh containing 1,848 elements and 954 nodes was generated.

Initially, the steady-state solution was obtained for the airfoil-alone configuration, and then the vortex was initialized at five chord lengths upstream from the airfoil leading edge. After initializing the vortex, a number of mesh adaptations were carried out to enhance the grid resolution for the vortex and for the flow around the airfoil. During the mesh adaptation, to prevent excessive division of small elements, the minimum size of tagged elemental faces was limited to 0.02.

Figure 6 (a) shows the initial mesh and the refined meshes after vortex initialization. The numbers of triangular elements of the refined meshes were 17064, 6494 and 4518 for DG(1), DG(2) and DG(3), respectively. In the case of DG(1), most of the refined elements were distributed around the airfoil and at the initialized vortex. However, for DG(2) and DG(3), mesh refinement was made mostly in the leading-edge and trailing-edge regions, and at the vortex. This is because the discretization error detected by the density difference was rapidly reduced for smooth flow around the airfoil as the order of accuracy increased. As shown in Fig. 6 (b), regardless of the accuracy of the scheme, the size of the element edges at the vortex core region after refinement was approximately 0.01, which is approximately a half of the vortex core radius. This is because minimum size constraint was forced to the refinement of the edges.

The unsteady time-accurate calculations were performed by marching the solution in time until the vortex travelled seven chord lengths downstream from the initialization. The mesh was dynamically adapted whenever the vortex travelled 0.005 chord lengths from the previous mesh adaptation. When measured at one chord length upstream of the airfoil leading edge, the vertical velocity of the vortex was diffused to 65%, 95%, and 98% of the initial value for DG(1), DG(2), and DG(3) calculations, respectively.

Figure 7 shows the instantaneous meshes and the corresponding pressure contours at several time levels. When the vortex was located ahead of the airfoil leading edge, small cell elements were concentrated at the vortex, at the airfoil leading edge, and near the trailing edge. At this time level, the number of elements for DG(1), DG(2), and DG(3) was 31888, 11849, and 7780, respectively. As the clockwise vortex collided to the airfoil and then travelled along the lower surface, an acoustic wave was generated from the airfoil leading edge due to the displacement of the stagnation point at the leading edge caused by the vortex collision. This wave subsequently propagated toward upstream of the flow field. In the case of DG(1) calculation, the vortex was much more diffused than that of higherorder calculations before collision, and thus a weaker acoustic wave was generated. The strength of BVI and acoustic wave propagation of DG(2) was similar to DG(3), even though decay of the DG(2) vortex in vertical velocity was slightly larger than that of DG(3) before collision. When the vortex travelled 6.25 chord lengths( $x_v$ =6.25), the number of elements increased to 52958, 29564, and 25208 for each calculation, mostly for capturing the acoustic wave propagation.

In Fig. 8, surface pressure histories are compared with measurement[28] and inviscid numerical simulations by other investigators[8,32] at three streamwise locations near the leading edge for both upper and lower surfaces. Ng and Hillier[32] performed the calculation by using a second-order upwind Godunov-type Euler solver on an extremely fine embedded structured grid. Tang and Baeder[8] performed the simulation by using an improved thirdorder accurate upwind method based on a piecewise quadratic reconstruction with a sixth-order compact difference scheme[29], coupled with a two-step grid redistribution method on structured grids. It is shown that on the upper surface the present results are in good agreements with the experimental data, even though the pressure peaks were predicted slightly upstream than measurement. This discrepancy was also observed in the predicted results by Ng and Hillier. On the lower surface, due to the lack of viscosity, relatively sharp peaks were observed in the present calculation, similar to other inviscid results [8, 32]. As the order of accuracy increased, the magnitude of the peaks also increased.

## 3.3 Transonic blade-vortex interaction

A transonic BVI problem was investigated for an NACA0012 airfoil section at a freestream Mach number of 0.8 and a blade-vortex offset of -0.26c, where c is the chord length. The vortex has a core radius of 0.05c and a non-dimensional strength of -0.2. The same initial unstructured mesh of the previous subsonic BVI studies was used.

A steady-state solution was initially obtained, and the vortex was initialized at five chord lengths upstream from the airfoil leading edge. After the vortex was initialized, a number of mesh adaptations were performed to enhance the grid resolution for the vortex and for flow around the airfoil as shown in Fig. 9. To suppress spurious oscillations near the shock wave and to prevent accuracy loss in the smooth flow field, a slope limiter[14] was applied to the elements tagged by the shock detector. Thus, the approximate solution of the tagged elements was represented by the polynomials of degree one, even for high-order calculations. By doing this, higherorder DGMs were stabilized in the presence of the shock wave without any deterioration of the solution accuracy in smooth flow regions. However, because of the limiting of the solution and the rather abrupt change of the order of accuracy for elements swept by the moving shock wave, spurious numerical noise was observed behind the shock wave, particularly for DG(2) and DG(3). This caused additional refinement of cells in the flow region behind the shock wave.

Figure 10 shows the instantaneous meshes and the corresponding pressure contours at several time levels as the vortex passes by the airfoil. When the vortex was located ahead of the airfoil leading edge, the effect of the vortex is relatively small, and the shock waves on the upper and lower surfaces of the airfoil were nearly symmetric. When the vortex was located underneath the airfoil, the shock wave on the lower surface strongly interacted with the vortex and finally bifurcated as the vortex passed by. The vortex restored its identity downstream of the airfoil trailing edge. During this transonic BVI, it was clearly observed that three acoustic waves were generated and propagated upstream. At first, a pressure wave was generated due to the movement of the leading edge stagnation point caused by the change of induced velocity by the vortex. Secondly, a compressibility wave was generated as a result of the interaction between the vortex and the shock wave on the airfoil lower surface. Finally, another compressibility wave was generated on the upper surface from the trailing edge when the vortex left the trailing edge. These complex flow features such as generation of acoustic waves, their migration to upstream, and the shock-wave bifurcation were well predicted using the present DGM solver coupled with a multi-level solution-adaptive dynamic mesh refinement/coarsening technique.

Figure 11 shows variation of the lift and quarterchord moment coefficients with the instantaneous streamwise position of the vortex during the interaction process. Lift was initially negative due to the downwash induced by the clockwise vortex located ahead of the leading edge. The lift reached a negative maximum when the vortex passed near the leading edge and then rapidly increased thereafter. A sudden sign change of moment from positive to negative was also observed as the vortex traveled below the airfoil. All three DG calculations show similar behaviors, and are in reasonable agreement with other Euler[2] and thin-layer Navier-Stokes[1] investigations based on a perturbation method.

## 4. Concluding Remarks

A high-order accurate flow solver based on a discontinuous Galerkin finite-element method has been developed for the accurate numerical simulation of blade-vortex interaction phenomena on unstructured meshes. The approximate solutions were expressed by truncated polynomial series consisted of the hierarchical basis functions based on Legendre polynomials. Roe's approximate

Riemann solver was applied to determine the interface flux. Time integration was achieved by using an explicit TVD Runge-Kutta method for unsteady time-dependent calculations, while an implicit time integration method based on the Euler backward differencing was used to obtain steadystate solutions. To suppress spurious oscillations near the shock wave and to prevent accuracy loss in smooth flow fields, a slope limiter was applied in conjuction with a shock detector. A multi-level solution-adaptive mesh refinement/coarsening technique was adopted to reduce the numerical error and to capture the vortex structure and the acoustic waves accurately.

A free vortex in freestream was investigated to assess the vortex-preserving property and the accuracy of the present flow solver. Then bladevortex interaction problems in subsonic and transonic freestreams were simulated. The results were compared with those of other numerical and experimental methods. It was shown that the present discontinuous Galerkin flow solver preserves the vortex structure for significantly longer vortex convection time and can accurately capture the generation and propagation of acoustic waves. It was found that the present high-order discontinuous Galerkin method coupled with a solution-adaptive mesh adaptation technique on unstructured meshes is a good alternative for accurately capturing complex flow features involving vortex convection.

## ACKNOWLEDGMENTS

This study has been supported by the KARI under KHP Dual-Use Component Development Program funded by the MOCIE.

## REFERENCES

- Srinivasan, G.R., McCroskey W.J., and Baeder J.D. Aerodynamics of two-dimensional blade-vortex interaction. *AIAA J*, 1986, **24**(10), pp 1569-1576.
- [2] Damodaran M. and Caughey D.A. Finite-volume calculation of inviscid transonic airfoil-vortex interaction. *AIAA J*, 1988, **26**(11), pp 1346-1353.
- [3] Steinhoff J. and Raviprakash G.K. Navier-Stokes computation of blade-vortex interaction using vorticity confinement. *AIAA Paper 95-0161*, 1995.
- [4] Rai M.M. Navier-Stokes simulation of blade-vortex interaction using high-order accurate upwind schemes. *AIAA-87-0543*, 1987.
- [5] Wake B.E and Choi D. Investigation of high-order upwinded differencing for vortex convection. *AIAA J*, 1996, **34**(2), pp 332-337.
- [6] Tang, L. and Baeder J.D. Time accurate Euler simulations of vortex convection, *Proceedings of the American Helicopter Society 52<sup>nd</sup> Annual Forum*, Washington, D.C., June, 1996, pp 1489-1499.

- [7] Tang, L. and Baeder J.D. Accurate Euler simulation of parallel blade-vortex interaction, *Proceedings of the American Helicopter Society* 53<sup>rd</sup> Annual Forum, Verginiar Beach, April, 1997, pp 708-718.
- [8] Tang, L. and Baeder J.D. Adaptive Euler simulations of airfoil-vortex interaction, *Int J Numer Meth Fluid*, 2007, **53**, pp 777-792.
- [9] Hwang C.J. and Kuo J.Y. Adaptive finite volume upwind approaches for aeroacoustic computations, *AIAA J*, 1997, **35**(8), pp 1286-1293.
- [10] Oh, W.S., Kim, J.S., and Kwon, O.J., Numerical simulation of two-dimensional blade-vortex interactions using unstructured adaptive meshes, *AIAA J*, 2002, 40, (3), pp 474-480.
- [11] Oh, W.S., Kim, J.S., and Kwon, O.J., Time-accurate Navier-Stokes simulation of vortex convection using an unstructured dynamic mesh procedure, *Comput Fluids*, 2003, **32**, (5), pp 727-749.
- [12] Cockburn, B. and Shu, C.W. The Runge-Kutta Discontinuous Galerkin Method for Conservation Laws V, *J Comput Phys*, 1998, **141**, pp 199-224.
- [13] Cockburn, B., Karniadakis, G., and Shu, C.W.(eds.) Discontinuous Galerkin Methods: Theory, Computation, and Applications, Lecture Notes in Computational Science and Engineering, Springer, New York, 1999.
- [14] Venkatakrishnan, V. Convergence to steady state solutions of the Euler equations on unstructured grids with limiters. *J Comput Phys*, 1995, **118**(1), pp 120-130.
- [15] Roe, P.L. Approximate Riemann solvers, parametric vectors, and difference schemes. *J Comput Phys*, 1881, **43**, pp 357-371.
- [16] Hartmann, R. Discontinuous Galerkin methods for compressible flows: higher order accuracy, error estimation and adaptivity. VKI LS 2006-01: CFD -Higher Order Discretization Methods, Belgium. 2005.
- [17] Gottlieb, S. and Shu, C.W. Total variation diminishing Runge-Kutta schemes, *Math Comput*, 1998, **67**, pp 73-85.
- [18] Solin, P., Segeth, P., and Zel, I. *Higher-Order Finite Element Methods,* Studies in Advanced Mathematics, Chapman and Hall, 2003.
- [19] Bassi, F. and Rebay, S. High-order accurate discontinuous finite element solution of the 2D Euler equations, *J Comput Phy*, 1997, **138**, pp 251-285.
- [20] Burbeau, A., Sagaut, P., and Bruneau, C.-H. A problem-independent limiter for high-order Runge-

Kutta discontinuous Galerkin methods. *J Comput Phys*, 2001, **169**, pp 111-150.

- [21] Hartmann R. and Houston P. Adaptive discontinuous Galerkin finite element methods for the compressible Euler equations. *J Comput Phy*, 2002; **183**, pp 508-532.
- [22] van der Vegt, J.J.W. and ven der Ven, H. Space-time discontinuous Galerkin finite element method with dynamic grid motion for inviscid compressible flows. J Comput Phys, 2002, 182, pp 546-585.
- [23] Xin, J. and Flaherty, J.E. Viscous stabilization of discontinuous Galerkin solutions of hyperbolic conservation laws. *Appl Numer Math*, 2006, 56, pp 444-458.
- [24] Luo, H., Baum, J.D., and Löhner, R. On the computation of steady-state compressible flows using a discontinuous Galerkin method, *Int J Numer Meth Eng*, 2008, **73**, pp 597-623.
- [25] Krivodonova, L., Xin, J., Remacle, J.F., Chevaugeon, N., and Flaherty, J.E. Shock detection and limiting with discontinuous Galerkin methods for hyperbolic conservation laws, *Appl Numer Math*, 2004, **48**, pp 323-338.
- [26] Qiu, J. and Shu, C.W. A comparison of troubled-cell indicators for Runge-Kutta discontinuous Galerkin methods using weighted essentially nonoscillatory limiters, *SIAM J Sci Comput*, 2005, **27**(3), pp 995-1013.
- [27] Sculley, M.P., Computation of helicopter rotor wake geometry and its influence on rotor harmonic loads, *Massachusetts Institute of Technology, ASRL TR-178-*1, 1975.
- [28] Lee, S. and Bershader, D. Head-on parallel bladevortex interaction, *AIAA J*, 1994, **32**, (1), pp 16-22.
- [29] Tang, L. and Baeder J.D. Improving Godunov-type reconstructions for simulation of vortex-dominated flows. *J Comput Phys*, 2006, **213**, pp 659-675.
- [30] Jiang, G.S. and Shu C.W., Efficient implementation of weighted ENO schemes, *J Comput Phys*, 1996, 126, pp 202-228.
- [31] Krist, S.L., Biedron, R.T., and Rumsey, C.L., CFL3D user's manual (version 5.0). NASA Langley Research Center, 1996.
- [32] Ng, N. and Hillier, R. Numerical investigation of the transonic blade-vortex interaction, *AIAA Paper 97-1846*, 1997.

Present calculations		Tang and Baeder[29]	
$(241 \times 41 \times 2$ uniform triangular mesh)		$(481 \times 81 \text{ uniform Cartesian mesh})$	
DG(4)	100%	Q6c	96%
DG(3)	98%	Q4c	91%
DG(2)	86%	WENO5	63%
DG(1)	53%	CFL3D	42%

Table 1 Normalized peak-to-peak tangential velocity  $\Delta v / \Delta v_0$  at x = 10 (200 $r_c$ ).



Fig. 1 Mapping  $F_{\kappa}$  of reference element  $\hat{K}$  to element K in real space.



Fig. 2 Hierarchical basis functions for DG(4),  $N_{\rm _{def}} = 10 \; . \label{eq:eq:entropy}$ 



Fig. 3 Unstructured mesh and tangential velocity profile of the initial vortex for free vortex convection problem.



(a) normalized peak-to-peak tangential velocity (b) normalized minimum density difference

Fig. 4 Decay of vortex convected in freestream.







Fig. 6 Refined unstructured meshes and initial vortex imposed in the steady flow field for the subsonic BVI problem.



Fig. 7 Instantaneous pressure contours and adapted meshes during subsonic BVI .



Fig. 8 Comparison of pressure histories at selected points near the leading edge for subsonic BVI.



Fig. 9 Refined meshes for initialized steady-state solutions.



(c) DG(3)

Fig. 10 Instantaneous pressure contours and adapted meshes during transonic BVI.



Fig. 11 Comparison of lift and moment variations with instantaneous vortex position for transonic BVI.