High-Order Implicit Large-Eddy Simulation for Transitional Aerodynamics Flows

by

Pablo Fernández

Ingeniero Industrial, Universidad de Valladolid (2013)

Submitted to the Department of Aeronautics and Astronautics in partial fulfillment of the requirements for the degree of

Master of Science

at the

MASSACHUSETTS INSTITUTE OF TECHNOLOGY

June 2016

© Massachusetts Institute of Technology 2016. All rights reserved.

Author .................................................................

Pablo Fernández

Department of Aeronautics and Astronautics

May 19, 2016

Certified by .................................................................

Jaime Peraire

H.N. Slater Professor of Aeronautics and Astronautics

Thesis Supervisor

Certified by .................................................................

Ngoc-Cuong Nguyen

Principal Research Scientist

Thesis Supervisor

Accepted by .................................................................

Paulo C. Lozano

Associate Professor of Aeronautics and Astronautics

Chair, Graduate Program Committee
High-Order Implicit Large-Eddy Simulation for Transitional Aerodynamics Flows

by

Pablo Fernández

Submitted to the Department of Aeronautics and Astronautics on May 19, 2016, in partial fulfillment of the requirements for the degree of Master of Science

Abstract

This thesis presents a high-order Implicit Large-Eddy Simulation (ILES) approach for simulating transitional aerodynamic flows. The approach consists of a hybridized Discontinuous Galerkin (DG) method for the discretization of the Navier-Stokes (NS) equations and a parallel preconditioned Newton-GMRES solver for the resulting nonlinear system of equations. The combination of hybridized DG methods with an efficient solution procedure leads to a high-order accurate NS solver that is competitive to alternative approaches, such as finite volume and finite difference codes, in terms of computational cost. The proposed approach is applied to transitional turbulent flows over a NACA 65-(18)10 compressor cascade and the Eppler 387 wing at Reynolds numbers up to 460,000. Grid convergence studies are presented and the required resolution to capture transition at different Reynolds numbers is investigated. Numerical results show rapid grid convergence and excellent agreement with experimental data. Focus is also placed on analyzing the structure of the boundary layer and the mechanism that causes transition to turbulence. Two-dimensional unstable modes in the form of Tollmien-Schlichting and Kevin-Helmholtz instabilities are found to be responsible for natural transition to turbulence through a laminar separation bubble. In short, this thesis aims to demonstrate the potential of high-order ILES for simulating transitional aerodynamic flows. This will be illustrated through numerical results and supported by theoretical considerations.

Thesis Supervisor: Jaime Peraire
Title: H.N. Slater Professor of Aeronautics and Astronautics

Thesis Supervisor: Ngoc-Cuong Nguyen
Title: Principal Research Scientist
Acknowledgments

Firstly, I would like thank to my advisors, Professor Jaime Peraire and Dr. Ngoc-Cuong Nguyen, for their continuous encouragement, support, and guidance during these two years. It has been a privilege to work with them and I cannot thank them enough for giving me the opportunity to come to MIT.

Secondly, I am grateful to Professor Mark Drela, Dr. Xevi Roca, Dr. Mani Sadeghi and Huan-Min Shang for their useful comments and suggestions, as well as to Dr. Chris Hill for providing with the computing resources to perform some of the simulations presented in this thesis.

I extend my gratitude to “la Caixa” Foundation for the Graduate Studies Fellowship that supported this work. I would also like to acknowledge Pratt & Whitney, the Air Force Office of Scientific Research, the Oak Ridge Leadership Computing Facility and the Argonne Leadership Computing Facility for partially supporting this effort.

I am thankful to my friends, in particular to my ACDL labmates, for cheering me up and helping me take a step back from research when necessary. I would also like to thank the faculty and staff in the Department of Aeronautics and Astronautics and the Center for Computational Engineering for their help and advice.

Last but clearly not least, I would like to thank my family, and in particular my parents, Antonio and Ana, for their patience and tremendous support during these two years. This thesis would just not have been possible without them.
# Contents

1 Introduction ........................................... 15
   1.1 Implicit Large-Eddy Simulation ..................... 16
   1.2 High-order discontinuous Galerkin methods ......... 17
   1.3 Scope and structure of the thesis .................. 18

2 Flow Discretization .................................... 21
   2.1 Preliminaries and notation .......................... 21
      2.1.1 Finite element mesh .......................... 21
      2.1.2 Finite element spaces ........................ 22
      2.1.3 Inner products ............................... 23
   2.2 Governing equations ................................ 23
   2.3 Spatial discretization: Hybridized DG methods .... 24
   2.4 Temporal discretization: Diagonally implicit Runge-Kutta methods . 28

3 Solution Method ....................................... 31
   3.1 Nonlinear solver .................................... 31
      3.1.1 Newton’s method ............................... 31
      3.1.2 Minimal residual algorithm for the initial guess .... 32
   3.2 Linear solver ...................................... 33
      3.2.1 Static condensation ............................ 33
      3.2.2 Generalized Minimal Residual method .......... 34
      3.2.3 Implementation of the matrix-vector product and BILU(0) solve 39
      3.2.4 Mixed-precision algorithm ..................... 39
### 4 Results

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.1 NACA 65-(18)10 compressor cascade</td>
<td>41</td>
</tr>
<tr>
<td>4.1.1 Details of the numerical discretization</td>
<td>42</td>
</tr>
<tr>
<td>4.1.2 Grid convergence study and comparison to second-order SGS-LES</td>
<td>42</td>
</tr>
<tr>
<td>4.1.3 Boundary layer analysis</td>
<td>44</td>
</tr>
<tr>
<td>4.2 Eppler 387 wing</td>
<td>51</td>
</tr>
<tr>
<td>4.2.1 Details of the numerical discretization</td>
<td>52</td>
</tr>
<tr>
<td>4.2.2 Grid convergence study and comparison with experimental data</td>
<td>53</td>
</tr>
<tr>
<td>4.2.3 Boundary layer analysis</td>
<td>55</td>
</tr>
</tbody>
</table>

### 5 Rationale for high-order ILES for transition prediction

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.1 Rationale for high-order methods</td>
<td>61</td>
</tr>
<tr>
<td>5.2 Rationale for ILES</td>
<td>63</td>
</tr>
</tbody>
</table>

### 6 Conclusions and future work

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.1 Conclusions</td>
<td>65</td>
</tr>
<tr>
<td>6.2 Future work</td>
<td>66</td>
</tr>
</tbody>
</table>

### A Boundary layer post-processing

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>A. Boundary layer post-processing</td>
<td>69</td>
</tr>
</tbody>
</table>
# List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2-1</td>
<td>Illustration of the degrees of freedom for the HDG method, the IEDG method, and the EDG method.</td>
<td>28</td>
</tr>
<tr>
<td>3-1</td>
<td>Illustration of the one-overlap decomposition for the HDG and IEDG schemes.</td>
<td>35</td>
</tr>
<tr>
<td>4-1</td>
<td>2D slice of Mesh No. 2 for the NACA 65-(18)10 compressor cascade.</td>
<td>43</td>
</tr>
<tr>
<td>4-2</td>
<td>Negative spanwise- and time-averaged pressure coefficient computed with high-order ILES and second-order FV SGS-LES for NACA 65-(18)10 in design condition. Iso-surface of the Q-criterion [26] colored by pressure is shown on the right.</td>
<td>44</td>
</tr>
<tr>
<td>4-3</td>
<td>Negative spanwise- and time-averaged pressure coefficient computed with high-order ILES and second-order FV SGS-LES for NACA 65-(18)10 in off-design condition. Iso-surface of the Q-criterion [26] colored by pressure is shown on the right.</td>
<td>44</td>
</tr>
<tr>
<td>4-4</td>
<td>Instantaneous (left) and time-averaged (right) velocity magnitude fields for NACA 65-(18)10 in design condition.</td>
<td>45</td>
</tr>
<tr>
<td>4-5</td>
<td>Streamwise displacement and momentum thickness (left) and shape parameter (right) along the suction side for NACA 65-(18)10 in design condition.</td>
<td>46</td>
</tr>
<tr>
<td>4-6</td>
<td>Amplitude of streamwise and cross-flow instabilities on the suction (left) and pressure (right) sides for NACA 65-(10) in design condition.</td>
<td>46</td>
</tr>
<tr>
<td>4-7</td>
<td>TS waves (left) and streamwise amplification factor (right) on the suction side for NACA 65-(18)10 in design condition.</td>
<td>47</td>
</tr>
</tbody>
</table>
4-8  Transition from TS to KH modes along the separated, suction side boundary layer for NACA 65-(18)10 in design (left) and off-design (right) conditions. ............................................. 48
4-9  Non-dimensional turbulent velocity profiles in the inner (left) and outer (right) layers at different locations along the suction side turbulent BL for NACA 65-(18)10 in design condition. .............................. 49
4-10 Instantaneous (left) and time-averaged (right) velocity magnitude fields for NACA 65-(18)10 in off-design condition. ................................. 49
4-11 Streamwise displacement and momentum thickness (left) and shape parameter (right) along the pressure side for NACA 65-(18)10 in off-design condition. .................................................. 50
4-12 Amplitude of streamwise and cross-flow instabilities (left) and TS waves (right) on the suction side for NACA 65-(10)18 in off-design condition. 50
4-13 Non-dimensional turbulent velocity profiles in the inner (left) and outer (right) layers at different locations along the suction side turbulent BL for NACA 65-(18)10 in off-design condition. .............................. 51
4-14 2D slice of Mesh No. 2 for the Eppler 387 wing. Tetrahedral meshes are created by subdividing each hexahedron into 6 tetrahedra. ........... 52
4-15 ILES prediction of the transitional flow over the Eppler 387 wing at $Re = 100,000$: Pressure coefficient (left), instantaneous spanwise velocity (top right), and iso-surface of the Q-criterion colored by pressure (bottom right). .................................................. 53
4-16 ILES prediction of the transitional flow over the Eppler 387 wing at $Re = 300,000$: Pressure coefficient (left), instantaneous spanwise velocity (top right), and iso-surface of the Q-criterion colored by pressure (bottom right). .................................................. 53
4-17 ILES prediction of the pressure coefficient (left) and skin friction coefficient (right) for the Eppler 387 wing at $Re = 460,000$. .......................... 54
4-18 Pressure coefficient near the leading edge for different sizes of the computational domain for the Eppler 387 wing at $Re = 100,000$. ......... 55
4-19 Amplitude of streamwise and cross-flow instabilities along the suction (left) and pressure (right) sides for the Eppler 387 wing at Reynolds number 100,000.

4-20 Amplitude of streamwise and cross-flow instabilities along the suction (left) and pressure (right) sides for the Eppler 387 wing at Reynolds number 300,000.

4-21 Spanwise- and time-averaged velocity magnitude field for the Eppler 387 wing at Reynolds number 100,000 (top), 300,000 (middle) and 460,000 (bottom).

4-22 Streamwise displacement and momentum thickness (left) and shape parameter (right) along the suction (top) and pressure (bottom) sides for the Eppler 387 wing at Reynolds number 100,000.

4-23 Streamwise displacement and momentum thickness (left) and shape parameter (right) along the suction side for the Eppler 387 wing at Reynolds number 300,000.
List of Tables

4.1 Details of the computational meshes considered for the NACA 65-(18)10 compressor cascade. 42
4.2 Details of the computational meshes considered for the Eppler 387 wing. 52
Chapter 1

Introduction

Fluid motion is governed by the Navier-Stokes equations. This set of time-dependent, nonlinear partial differential equations (PDEs) exhibits chaotic behavior when the Reynolds number is beyond some threshold. Under these conditions, the fluid flow undergoes transition from laminar to turbulent regime; which results in a dramatic change in the transport properties and overall behavior of the flow field.

As it happens, a flow may be fully laminar, fully turbulent, or undergo transition depending on applications. The latter regime is usually encountered in aerodynamic flows with Reynolds numbers between 30,000 and 600,000, and will be referred to as \textit{transitional flows} or \textit{transitional turbulent flows} throughout this thesis. In this regime, the transition location has a very significant impact on the aerodynamic performance, and its accurate prediction is key for engineering purposes. This range of Reynolds numbers is present in a growing number of applications, including hypersonic flows (reentry vehicles, scramjets, missiles), turbomachinery flows and flying animal locomotion, to name a few. Perhaps most importantly, transitional flows are often encountered in Unmanned Air Vehicles (UAVs), commonly known as drones. Boosted by its commercial and civilian side, the UAV industry has experienced very rapid growth in the last years and this trend is expected to continue in the years to come. In this context, accurate and reliable simulation techniques for transitional flows are much-needed to assist the design process.
1.1 Implicit Large-Eddy Simulation

Numerical simulation of transitional turbulent flows is a challenging problem. First, turbulent flows exhibit spatial and temporal scales that differ by several orders of magnitude, thereby making numerical simulation of turbulence computationally demanding. Second, transitional flows are hard to predict due to the complex physical phenomena that take place when the flow undergoes transition from laminar to turbulent regime. Indeed, transitional turbulent flows are not yet well understood theoretically despite being studied for over a century.

Complete information about turbulent flows can be obtained by means of direct numerical simulation (DNS). The idea behind DNS is to employ a spatial and temporal discretization with sufficient resolution to represent all the scales present in the turbulent flow; which yields a computational cost $\mathcal{O}(Re^{11/4})$ in the limit of large Reynolds numbers. Hence, despite the availability of powerful supercomputers, DNS remains intractable for most practical applications.

Large-Eddy Simulation (LES) is a viable alternative to DNS. The central premise of LES is that large-scale eddies dominate the turbulent transport and energy budget, so that a numerical simulation will provide a realistic depiction of the flow if it captures those scales explicitly and somehow accounts for the small scales that cannot be resolved. Also, the small scales tend to be more homogeneous and isotropic, and hence easier to model. Strategies for dealing with the small turbulent scales include explicit subgrid-scale (SGS) modeling and implicit numerical dissipation.

In the classical (explicit) LES approach, the large-scale eddies of the flow field are resolved and the small scales are modeled using a SGS model. The development of SGS models has been a subject of intense interest for decades, in particular in the 1990s [20,27,36]. It turns out, however, that the leading-order term of the truncation error introduced by many numerical schemes is similar in form and magnitude to conventional SGS models. As a rule of thumb, a stable numerical scheme often achieves stability by introducing truncation errors that replicate the effect of the subgrid scales into the resolved scales; which corresponds to dissipation in under-
resolved turbulent simulations. This has been known to be the case for monotone (i.e. stable) numerical schemes for about 20 years [3, 9, 15, 22, 32, 50] and is now also thought to be the case for stabilized high-order methods [19, 37].

A natural alternative to the classical LES approach is therefore to use the numerical dissipation of the discretization scheme to account for the dissipation that takes place in the unresolved scales, leading to the so-called Implicit LES (ILES). The ILES approach was first introduced in 1990 by Boris et al. [2], and has been successfully applied with a number of different schemes, including finite volume methods [14, 16, 17], standard [21] and compact [18, 55] finite difference methods, spectral difference methods [59], spectral/hp element methods [29], flux reconstruction methods [45], and discontinuous Galerkin methods [13, 39, 49, 56–58]. ILES benefits from its easy implementation without a SGS model and currently gains considerable attention from researchers in the computational fluid dynamics community. As pointed out by Spalart [54], this may be attributed to the fact that research has failed to show an advantage of sophisticated SGS models over the same-cost LES with a simplistic model –or even with no model– and a slightly finer grid.

1.2 High-order discontinuous Galerkin methods

While second-order finite volume (FV) schemes have been widely used in academia and industry for LES, transitional flows do share many features with wave propagation phenomena for which high-order accuracy is known to be key. Let us consider natural transition to turbulence. The main difficulty to numerically capture transition is the very small magnitude of the perturbations that get exponentially amplified along the unstable portion of the laminar boundary layer. These small perturbations are ultimately responsible for the so-called nonlinear breakdown and transition to turbulence. The amplitude of these instabilities at the location in which the boundary layer becomes unstable is up to ten orders of magnitude below the freestream velocity [53]. As a result, very small amount of numerical dissipation and dispersion is needed to capture them and accurately predict the transition location. Overdissipation of
low-order schemes may kill these small perturbations and lead to inaccurate transition prediction. This motivates researchers to consider high-order methods for the simulation of transitional flows.

High-order finite difference (FD), finite volume and finite element (FE) methods have been developed for numerical simulation of turbulent flows. Among them, we rely on Discontinuous Galerkin (DG) finite element methods for several reasons. First, DG methods are based on a strong mathematical foundation that can be exploited for error estimation and mesh adaptation purposes. Also, they provide local conservation, a stable discretization of the convective operator, and are well-suited for turbulent simulations due to the \textit{ab initio} separation of scales in the variational formulation. Most importantly, DG methods allow for high-order implementations on complex geometries and unstructured meshes. This is crucial for engineering purposes due to the need to simulate complex three-dimensional geometries. On the other hand, DG methods require significantly more operations per computational cell than FD and FV methods. Hence, the need for more computationally efficient DG methods becomes apparent. In the spirit of making DG methods competitive, researchers have recently developed the Hybridizable DG (HDG) method [5, 41, 41–43, 46] and the Embedded DG (EDG) method [5, 6, 47]. These DG methods result in spatial discretizations that have less globally coupled unknowns than other DG methods, thereby reducing the computational cost and memory footprint.

1.3 Scope and structure of the thesis

At present, ILES of transitional turbulent flows using high-order DG methods is limited to Reynolds numbers of 100,000 or less [13, 39, 49, 56–58]. It may be attributed to the fact that higher Reynolds number flows would require significantly more computational effort than current DG methods could afford. This motivates us to employ hybridized DG methods [44], which generalize the HDG and the EDG methods, for ILES of transitional turbulent flows. This thesis contains novel contributions from the points of view of methodology and applications. In terms of methodology, we
propose a parallel preconditioned Newton-GMRES solver for the nonlinear system arising from the hybridized DG discretization of the Navier-Stokes equations. In terms of applications, this thesis is, together with preliminary results in [10,11], the first attempt at high-order DG ILES of transitional flows at Reynolds number up to 460,000 that is far beyond the existing simulations in the literature. We believe that this work will hold a promising approach to simulate transitional flows and contribute to the ongoing research in ILES of turbulent flows.

The thesis is organized as follows. In Chapter 2, we describe the numerical discretization of the Navier-Stokes equations using hybridized DG methods. In Chapter 3, we discuss solution methods for the nonlinear system of equations arising from the numerical discretization described in Chapter 2. In Chapter 4, the proposed approach is applied to the ILES of transitional flows over a NACA 65-(18)10 compressor cascade and the Eppler 387 wing at Reynolds numbers ranging from 100,000 to 460,000. A rationale for the success of high-order ILES for transition prediction is presented in Chapter 5. Finally, some conclusions and future work are discussed in Chapter 6. The discussion herein will focus on highlighting the key points and contributions of this work, while providing the interested reader with a number of references with further details. We hope this will avoid unnecessary complexity and allow for a more dynamic and informative exposition.
Chapter 2

Flow Discretization

This chapter presents an overview of hybridized discontinuous Galerkin methods for solving the unsteady, compressible Navier-Stokes equations. For a thorough discussion on the stability, accuracy order and computational efficiency of this family of schemes, the interested reader is referred to dedicated papers on these methods [5, 6, 40–44, 46, 47].

2.1 Preliminaries and notation

2.1.1 Finite element mesh

Let \( \Omega \subseteq \mathbb{R}^d \) with \( d = 3 \) be a physical domain with Lipschitz boundary \( \partial \Omega \). We denote by \( \mathcal{T}_h \) a collection of disjoint, regular, \( p \)-th degree curved elements \( K \) that partition \( \Omega \), and set \( \partial \mathcal{T}_h := \{ \partial K : K \in \mathcal{T}_h \} \) to be the collection of the boundaries of the elements in \( \mathcal{T}_h \). For an element \( K \) of the collection \( \mathcal{T}_h \), \( F = \partial K \cap \partial \Omega \) is a boundary face if its \( d-1 \) Lebesgue measure is nonzero. For two elements \( K^+ \) and \( K^- \) of \( \mathcal{T}_h \), \( F = \partial K^+ \cap \partial K^- \) is the interior face between \( K^+ \) and \( K^- \) if its \( d-1 \) Lebesgue measure is nonzero. We denote by \( \mathcal{E}_h^I \) and \( \mathcal{E}_h^B \) the set of interior and boundary faces, respectively, and we define \( \mathcal{E}_h := \mathcal{E}_h^I \cup \mathcal{E}_h^B \) to be the union of interior and boundary faces. Note that, by definition, \( \partial \mathcal{T}_h \) and \( \mathcal{E}_h \) are different. More precisely, an interior face is counted twice in \( \partial \mathcal{T}_h \) but only once in \( \mathcal{E}_h \), whereas a boundary face is counted
once both in $\partial \mathcal{T}_h$ and $\mathcal{E}_h$.

### 2.1.2 Finite element spaces

Let $\mathcal{P}_k(D)$ denote the space of complete polynomials of degree $k$ on a domain $D \in \mathbb{R}^n$, let $L^2(D)$ be the space of square-integrable functions on $D$, and let $\psi^p_K$ denote the $p$-th degree parametric mapping from the reference element $K_{\text{ref}}$ to some physical element $K \in \mathcal{T}_h$. We then introduce the following discontinuous finite element spaces:

\begin{align}
\mathcal{Q}_h^k &= \{ \mathbf{r} \in [L^2(\mathcal{T}_h)]^{m \times d} : (\mathbf{r} \circ \psi^p)|_K \in [\mathcal{P}_k(K_{\text{ref}})]^{m \times d} \ \forall K \in \mathcal{T}_h \}, \quad (2.1a) \\
\mathcal{V}_h^k &= \{ \mathbf{w} \in [L^2(\mathcal{T}_h)]^m : (\mathbf{w} \circ \psi^p)|_K \in [\mathcal{P}_k(K_{\text{ref}})]^m \ \forall K \in \mathcal{T}_h \}, \quad (2.1b)
\end{align}

where $m$ denotes the number of equations of the conservation law, i.e. $m = d + 2$ for the Navier-Stokes system. The subscript $h$ follows the usual convention that indicates that the space is finite dimensional and associated to a certain discretization of characteristic element size $h$.

Let us also denote the $p$-th degree parametric mapping from the reference face $F_{\text{ref}}$ to some physical face $F$ by $\phi^p_F$. We then introduce the following traced finite element spaces on the mesh skeleton $\mathcal{E}_h$:

\begin{align}
\mathcal{\tilde{M}}_h^k &= \{ \mu \in [L^2(\mathcal{E}_h)]^m : (\mu \circ \phi^p_F)|_F \in [\mathcal{P}_k(F_{\text{ref}})]^m \ \forall F \in \mathcal{E}_h \}, \quad (2.2a) \\
\mathcal{\hat{M}}_h^k &= \{ \mu \in [C^0(\mathcal{E}_h)]^m : (\mu \circ \phi^p_F)|_F \in [\mathcal{P}_k(F_{\text{ref}})]^m \ \forall F \in \mathcal{E}_h \}, \quad (2.2b)
\end{align}

where $C^0(D)$ is the space of continuous functions on $D$. Note that $\mathcal{\tilde{M}}_h^k$ consists of functions which are discontinuous at the boundaries of the faces, whereas $\mathcal{\hat{M}}_h^k$ consists of functions that are continuous at the boundaries of the faces. We also denote by $\mathcal{M}_h^k$ a traced finite element space that satisfies $\mathcal{\tilde{M}}_h^k \subseteq \mathcal{M}_h^k \subseteq \mathcal{\hat{M}}_h^k$. In particular, we define

\begin{align}
\mathcal{M}_h^k &= \{ \mu \in [L^2(\mathcal{E}_h)]^m : (\mu \circ \phi^p_F)|_F \in [\mathcal{P}_k(F_{\text{ref}})]^m \ \forall F \in \mathcal{E}_h, \ \mu|_{\mathcal{E}_h^E} \in [C^0(\mathcal{E}_h^E)]^m \}, \quad (2.3)
\end{align}
where \( \mathcal{E}_h^E \) is a subset of \( \mathcal{E}_h \). Note that \( \mathcal{M}_h^k \) consists of functions which are continuous on \( \mathcal{E}_h^E \) and discontinuous on \( \mathcal{E}_h^H := \mathcal{E}_h \setminus \mathcal{E}_h^E \). Furthermore, if \( \mathcal{E}_h^E = \emptyset \) then \( \mathcal{M}_h^k = \mathcal{\tilde{M}}_h^k \), and if \( \mathcal{E}_h^E = \mathcal{E}_h \) then \( \mathcal{M}_h^k = \mathcal{M}_h^k \). Different choices of \( \mathcal{E}_h^E \) lead to different discretization methods that have different properties in terms of convergence rates, stability, and number of globally coupled unknowns. We shall discuss various rational choices below.

### 2.1.3 Inner products

In order to express the hybridized DG discretization of the Navier-Stokes equations in a compact way, we define several inner products associated to the previous finite element spaces. In particular, given \( w, v \in \mathcal{V}_h^k \), \( W, V \in \mathcal{Q}_h^k \) and \( \eta, \zeta \in \mathcal{M}_h^k \), we write

\[
(w, v)_{\mathcal{T}_h} = \sum_{K \in \mathcal{T}_h} (w, v)_K = \sum_{K \in \mathcal{T}_h} \int_K w \cdot v_K, \tag{2.4a}
\]

\[
(W, V)_{\mathcal{T}_h} = \sum_{K \in \mathcal{T}_h} (W, V)_K = \sum_{K \in \mathcal{T}_h} \int_K W : V, \tag{2.4b}
\]

\[
\langle \eta, \zeta \rangle_{\partial \mathcal{T}_h} = \sum_{K \in \mathcal{T}_h} \langle \eta, \zeta \rangle_{\partial K} = \sum_{K \in \mathcal{T}_h} \int_{\partial K} \eta \cdot \zeta, \tag{2.4c}
\]

where : denotes the Frobenius inner product \( W : V = \text{tr}(W^T V) \).

### 2.2 Governing equations

We consider the unsteady, compressible Navier-Stokes equations written in non-dimensional conservation form as

\[
\begin{align*}
q - \nabla u &= 0, \quad \text{in } \Omega \times (0, T), \\
\frac{\partial u}{\partial t} + \nabla \cdot F(u, q) &= 0, \quad \text{in } \Omega \times (0, T).
\end{align*}
\tag{2.5}
\]
Here, \( u = (\rho, \rho v_j, \rho E), \ j = 1, \ldots, d \) is the \( m \)-dimensional vector of non-dimensional conserved quantities, and \( F(u, q) \) are the Navier-Stokes fluxes of dimension \( m \times d \)

\[
F(u, q) = \begin{pmatrix}
\rho v_j \\
\rho v_j v_i + \delta_{ij} p \\
v_j (\rho E + p)
\end{pmatrix} - \begin{pmatrix}
0 \\
\tau_{ij} \\
v_i \tau_{ij} + f_j
\end{pmatrix}.
\]

(2.6)

For a Newtonian, calorically perfect gas in thermodynamic equilibrium, the non-dimensional viscous stress tensor, heat flux, and pressure are given by

\[
\tau_{ij} = \frac{1}{Re} \left[ \left( \frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} \right) - \frac{2}{3} \frac{\partial v_k}{\partial x_k} \delta_{ij} \right],
\]

(2.7a)

\[
f_j = -\frac{\gamma}{Re \ Pr} \frac{\partial T}{\partial x_j},
\]

(2.7b)

\[
p = (\gamma - 1) \rho \left( E - \frac{1}{2} v_k v_k \right),
\]

(2.7c)

respectively. The Stokes’ hypothesis for the second coefficient of viscosity and the Fourier’s laws of thermal conduction have been applied to obtain the constitutive relations above. Also, \( Re \) denotes the freestream Reynolds number, \( Pr \) the Prandtl number, and \( \gamma \) the specific heat ratio. In particular, \( Pr = 0.72 \) and \( \gamma = 1.4 \) for air. The freestream Mach number \( M_\infty \) enters in the Navier-Stokes system through the non-dimensional freestream pressure \( p_\infty = 1/(\gamma M_\infty^2) \).

### 2.3 Spatial discretization: Hybridized DG methods

The hybridized DG discretization [44] of the governing equations (2.5) reads as follows:

Find \( (q_h(t), u_h(t), \hat{u}_h(t)) \in \mathcal{Q}_h^k \times \mathcal{V}_h^k \times \mathcal{M}_h^k \) such that

\[
\left( q_h, r \right)_{\mathcal{T}_h} + \left( u_h, \nabla \cdot r \right)_{\mathcal{T}_h} - \left( \hat{u}_h, r \cdot n \right)_{\partial \mathcal{T}_h} = 0,
\]

(2.8a)

\[
\left( \frac{\partial u_h}{\partial t}, w \right)_{\mathcal{T}_h} - \left( F(u_h, q_h), \nabla w \right)_{\mathcal{T}_h} + \left( \dot{f}_h(\hat{u}_h, u_h, q_h), w \right)_{\partial \mathcal{T}_h} = 0,
\]

(2.8b)

\[
\left( \dot{f}_h(\hat{u}_h, u_h, q_h), \mu \right)_{\partial \mathcal{T}_h \setminus \partial \Omega} + \left( \dot{b}_h(\hat{u}_h, u_h, q_h), \mu \right)_{\partial \Omega} = 0,
\]

(2.8c)
for all \((r, w, \mu) \in \mathcal{Q}_h^k \times \mathcal{V}_h^k \times \mathcal{M}_h^k\) and all \(t \in (0, T)\). Furthermore, \((u_h(t = 0) - u_0, r)_{\tau_h} = 0\), where \(u_0\) is the initial condition. Due to ergodicity, almost any\footnote{We use the term almost any here in the conventional sense in measure theory. This is, to indicate that the set of initial conditions that lead to a different time-average behavior is a set of measure zero.} initial condition leads to the same time-average behavior of the system. The particular choice of \(u_0\), however, does affect the run-up time that it takes to reach the attractor of the system. A strategy to reduce the run-up time will be presented below.

Equations (2.8a)-(2.8b) weakly impose the Navier-Stokes equations, whereas Eq. (2.8c) weakly enforces the flux conservation across elements and the boundary conditions. The introduction of the solution trace \(\hat{u}_h\) as an additional unknown is characteristic of hybridized DG methods and allows to parameterize the solution inside the elements \((q_h, u_h)\) as a function of \(\hat{u}_h\). This in turn yields a reduced number of globally coupled unknowns compared to other DG methods. Also, \(n\) in Eq. (2.8a) is the unit normal vector pointing outwards the elements and \(\hat{b}_h\) is the boundary flux, whose precise definition depends on the type of boundary condition. For the simulations presented in this thesis, the following approximate non-reflecting boundary condition is imposed at the inlet and outlet sections of the flow field:

\[
\hat{b}_h = \frac{1}{2}(A_n + |A_n|) \cdot (\hat{u}_h - u_h) + \frac{1}{2}(A_n - |A_n|) \cdot (\hat{u}_h - u_h),
\]

(2.9)

where \(u_b\) is a given boundary state, and \(A_n = \partial(F_{\text{inv}} \cdot n) / \partial u = X A_n X^{-1}\) denotes the Jacobian of the inviscid normal flux to the boundary evaluated at \(\hat{u}_h\). Also, \(|A_n| := X|A_n|X^{-1}\). This boundary condition corresponds, up to first order, to choosing the incoming Riemann invariants from the boundary state \(u_b\) and extrapolating the outgoing Riemann invariants from the interior of the domain. A non-slip, adiabatic wall boundary condition is used on the airfoil surface, and periodicity is imposed on the remaining boundaries of the domain.

Also, \(\hat{f}_h\) in Eq. (2.8b)-(2.8c) denotes the numerical flux defined as

\[
\hat{f}_h(\hat{u}_h, u_h, q_h) = F(\hat{u}_h, q_h) \cdot n + S(\hat{u}_h, u_h) \cdot (u_h - \hat{u}_h).
\]

(2.10)
$S \in \mathbb{R}^{m \times m}$ is the so-called stabilization tensor and plays an important role in the accuracy and stability of the resulting scheme. For convection-dominated flows, it is customary to define $S$ to stabilize the convective operator as that suffices in practice to stabilize the diffusive operator as well. In particular, let us consider the following choices for the stabilization tensor

\begin{align}
S &= \tau I_{m \times m}, \quad (2.11a) \\
S &= |A_n| = X|\Lambda_n|X^{-1}, \quad (2.11b) \\
S &= \frac{1}{2} (A_n + |A_n|) = \frac{1}{2} X (\Lambda_n + |\Lambda_n|) X^{-1}, \quad (2.11c)
\end{align}

where $I_{m \times m}$ denotes the $m \times m$ identity matrix and $\tau \in \mathbb{R}$. Equation (2.11a) is widely used in the HDG community. In particular, $\tau = \lambda_{max}(\hat{u}_h) = \max_{i=1,...,m} |\lambda_i(\hat{u}_h)|$ in a Lax-Friedrichs manner provides a systematic way to ensure linear stability of the convective operator. This choice, however, is problematic in the incompressible limit since the energy equation and $\lambda_{max}$ scale as $\Theta(1/M^2_\infty)$, whereas the continuity and momentum equations are $\Theta(1)$. The hybridized DG method then degenerates into the continuous Galerkin (CG) method for the continuity and momentum equations, hence inheriting the well-known stability issues of the CG method. This can be addressed by an approach based on characteristic variables, such as Equations (2.11b) and (2.11c); which properly stabilize each component of the system even in the incompressible limit. Furthermore, they yield a smaller discarded fill in the incomplete LU factorization to be described in Chapter 4, hence improving the preconditioner approximation. Despite the previous observations, Eq. (2.11a) is used in this thesis due to the lack of a comparative study between these choices when the numerical simulations were performed. In particular, we define $\tau = \lambda_{max}(u_\infty)$ to reduce the nonlinearity of the approach.

**On the choice of the initial condition $u_0$**

We exploit the quasi-2D nature of the flows considered in this work to inexpensively compute a convenient initial condition. In particular, we solve the two-dimensional,
unsteady Navier-Stokes equations in a 2D slice of the domain until the attractor of
the two-dimensional NS system is reached. This state is close in phase space to the
three-dimensional NS attactor, so that its use as initial conduction reduces the run-up
time, and thus total number of time steps to converge the statistics of the turbulent
flow.

On the choice of the subset $\mathcal{E}_h^E$

It remains to analyze the choices for $\mathcal{E}_h^E$ in the definition of the space $\mathcal{M}_h^k$. Let
us consider two simple choices. The first one is $\mathcal{E}_h^E = \emptyset$ and yields $\mathcal{M}_h^k = \widetilde{\mathcal{M}}_h^k$.
This corresponds to the Hybridizable Discontinuous Galerkin (HDG) method [5, 40–
43, 46]. The second choice is $\mathcal{E}_h^E = \mathcal{E}_h$ and implies $\mathcal{M}_h^k = \overline{\mathcal{M}}_h^k$. This corresponds
to the Embedded Discontinuous Galerkin (EDG) method [5, 6, 47] and makes the
approximation space $\mathcal{M}_h^k$ continuous over $\mathcal{E}_h$. Hence, the HDG method and the EDG
method can be considered as two particular instances of a larger family of methods,
that we refer to as hybridized Discontinuous Galerkin methods. In particular, the
only difference between HDG and EDG lies in the definition of the approximation
space $\mathcal{M}_h^k$ for the trace of the solution. This subtlety is responsible for important
differences between both methods in terms of accuracy, stability, and computational
efficiency.

Another interesting choice of the approximation space $\mathcal{M}_h^k$ is obtained by setting
$\mathcal{E}_h^E = \mathcal{E}_h^I$. This implies $\overline{\mathcal{M}}_h^k \subset \mathcal{M}_h^k \subset \widetilde{\mathcal{M}}_h^k$, and the resulting approximation
space consists of functions that are continuous everywhere but at the borders of
the boundary faces. The resulting method has an HDG flavor on the boundary faces
and an EDG flavor on the interior faces, and is therefore referred to as the Interior
Embedded DG (IEDG) method [44]. Figure 2-1 illustrates the degrees of freedom for
the HDG method, the IEDG method, and the EDG method in a four-element mesh.
The blue nodes represent the degrees of freedom of the approximate solution $(u_h, q_h)$,
while the red nodes represent the degrees of freedom of the approximate trace $\tilde{u}_h$.
Again, the three schemes differ from each other only in the degrees of freedom of the
approximate trace.
We note that the IEDG method enjoys advantages of both HDG and EDG. First, IEDG inherits the reduced number of global degrees of freedom, and thus the computational efficiency, of EDG. In fact, the degrees of freedom of the approximate trace on $\mathcal{E}_h^B$ can be locally eliminated without affecting the sparsity pattern of the Jacobian matrix; which yields an even smaller number of global degrees of freedom than in the EDG method. Second, the IEDG scheme enforces the boundary conditions as strongly as the HDG method, hence retaining most of the stability and robustness of HDG. These features make the IEDG method an excellent alternative to the HDG and EDG schemes.

![Illustration of the degrees of freedom for the HDG method, the IEDG method, and the EDG method.](image)

Figure 2-1: Illustration of the degrees of freedom for the HDG method, the IEDG method, and the EDG method.

### 2.4 Temporal discretization: Diagonally implicit Runge-Kutta methods

The semi-discrete system (2.8a-2.8c) is further discretized in time using $L$-stable, diagonally implicit Runge-Kutta (DIRK) schemes [1]. The application of a DIRK($q,r$) scheme, where $q$ is the number of stages and $r$ is the order of accuracy, to Equations (2.8a-2.8c) can be recasted as a sequence of $q$ nonlinear systems of equations at every
time step of the form

\[ (q_h, r)_{T_h} + (u_h, \nabla \cdot r)_{T_h} - \langle \tilde{u}_h, r \cdot n \rangle_{\partial T_h} = 0, \quad (2.12a) \]

\[ (Cu_h, w)_{T_h} - (F(u_h, q_h), \nabla w)_{T_h} + \left\langle \hat{f}_h(\tilde{u}_h, u_h, q_h), w \right\rangle_{\partial T_h} = s, \quad (2.12b) \]

\[ \left\langle \hat{f}_h(\tilde{u}_h, u_h, q_h), \mu \right\rangle_{\partial T_h \setminus \partial \Omega} + \left\langle \hat{b}_h(\tilde{u}_h, u_h, q_h), \mu \right\rangle_{\partial \Omega} = 0, \quad (2.12c) \]

for all \((r, w, \mu) \in Q_h^k \times \mathcal{V}_h^k \times \mathcal{M}_h^k\). \(C \in \mathbb{R}\) is a function of the time-step size \(\Delta t\), the particular DIRK\((q,r)\) scheme, and the current DIRK stage within the current time-step. Also, \(s\) is a source term that depends on the solution at the previous DIRK stages.

We note that the use of \(L\)-stable schemes allows for choosing the time-step size based on physical considerations instead of on numerical stability issues. This is particularly advantageous for transition prediction, as discussed later in Chapter 5.
Chapter 3

Solution Method

In this chapter, we propose a parallel Newton-GMRES method for the solution of the nonlinear system of equations (2.12a-2.12c) arising from the spatial and temporal discretization of the Navier-Stokes equations.

3.1 Nonlinear solver

3.1.1 Newton’s method

Let us denote \( z_h = (q_h, u_h) \in Q_h^k \times V_h^k \). At any given step \( n = q(\ell - 1) + s \) of a DIRK\((q, r)\) scheme \( (q \) is the number of stages, \( r \) is the order of accuracy, \( \ell \) is the current time step, and \( s = 1, ..., q \) is the current stage within the current time step), the nonlinear system of equations (2.12a-2.12c) can be written as

\[
R_{CL}(z_h^n, \hat{u}_h^n) = 0, \tag{3.1a}
\]

\[
R_{FC}(z_h^n, \hat{u}_h^n) = 0, \tag{3.1b}
\]

where \( R_{CL} \) and \( R_{FC} \) are the discrete nonlinear residuals associated to Eq. (2.12a)-(2.12b) and Eq. (2.12c), respectively. Here, CL stands for Conservation Law and FC stands for Flux Conservation.

The Newton’s method is used to solve the nonlinear system (3.1). Linearizing
(3.1) around the solution \((z_h^{m,n}, \hat{u}_h^{n,m})\) at the Newton step \(m = 0, 1, \ldots\), we arrive at the following linear system

\[
\begin{bmatrix}
A^{n,m} & B^{n,m} \\
C^{n,m} & D^{n,m}
\end{bmatrix}
\begin{bmatrix}
\delta Z^{n,m} \\
\delta \hat{U}^{n,m}
\end{bmatrix}
= 
\begin{bmatrix}
F^{n,m} \\
G^{n,m}
\end{bmatrix},
\]

(3.2)

where \(\delta Z^{n,m}\) is the vector of degrees of freedom for \(\delta z_h^{m,n} = (\delta q_h^{n,m}, \delta u_h^{n,m}) \in \mathbb{Q}_h \times \mathbb{V}_h\), and \(\delta \hat{U}^{n,m}\) is the vector of degrees of freedom for \(\delta \hat{u}_h^{n,m} \in \mathbb{M}_h\). The next Newton iterate is defined as \((z_h^{n,m+1}, \hat{u}_h^{n,m+1}) := (z_h^{n,m} + \delta z_h^{n,m}, \hat{u}_h^{n,m} + \delta \hat{u}_h^{n,m})\), and the process is repeated until the norm of the full residual vector \(R_{NS} := (R_{CL}, R_{FC})\) is less than a specified tolerance.

### 3.1.2 Minimal residual algorithm for the initial guess

The convergence of Newton’s method depends on the initial guess \((z_h^{n,0}, \hat{u}_h^{n,0})\). In particular, Newton’s method can be shown to display quadratic convergence for smooth functions provided the initial guess is sufficiently close to the root.

One common initialization strategy is to take the initial guess to be the solution at the previous DIRK step, namely, \((z_h^{n,0}, \hat{u}_h^{n,0}) := (z_h^{n-1}, \hat{u}_h^{n-1})\). In order to reduce the number of Newton iterations relative to this simple initialization strategy, we propose a minimal residual algorithm to compute a better initial guess. In particular, we express the initial guess as a linear combination of the solutions at the \(J\) previous DIRK steps, namely, \((z_h^{n,0}, \hat{u}_h^{n,0}) := \sum_{j=1}^{J} \alpha_j (z_h^{n-j}, \hat{u}_h^{n-j})\). The coefficients \(\alpha_j\) are found as the minimizer of the following nonlinear least squares problem

\[
(\alpha_1, \ldots, \alpha_J) = \arg \min_{(\beta_1, \ldots, \beta_J) \in \mathbb{R}^{J}} \left\| R_{NS} \left( \sum_{j=1}^{J} \beta_j (z_h^{n-j}, \hat{u}_h^{n-j}) \right) \right\|.
\]

(3.3)

This optimization problem is solved by using the Levenberg–Marquardt (LM) algorithm [28, 33]. The LM algorithm requires us to compute the gradient vectors \(\partial R_{NS}/\partial \beta_j\); which are approximated by finite differences. This in turn requires a small number of residual evaluations. Since a residual evaluation is more than two
orders of magnitude faster than the Jacobian computation and the linear solve, this minimal residual procedure adds very little to the overall computational cost of the solution method. Also, $\sum_{j=1}^{J} \alpha_j = 1$ could be included as an explicit constraint in Eq. (3.3) in order to improve the robustness of the approach. In practice, however, we have found no differences between solving the constrained and unconstrained versions of the problem as non-physical states are usually far from the optimal solution.

### 3.2 Linear solver

We describe next our approach for solving the linear system (3.2). Hereinafter, we shall drop the superscript $n, m$ to simplify the notation.

#### 3.2.1 Static condensation

Due to the discontinuous nature of the finite element spaces, the matrix $A$ has block-diagonal structure. This allows us to inexpensively compute its inverse and eliminate $\delta Z$ to obtain a smaller linear system in terms of $\delta \hat{U}$ only as follows

$$K \delta \hat{U} = R,$$

where $K = D - C A^{-1}B$ and $R = G - C A^{-1}F$.

This is the global system to be solved at every Newton iteration. One of the advantages of the hybridized DG methods lies precisely in the fact that the size and number of nonzeros of $K$ are much smaller than those of the global matrix in other DG methods. This is in turn due to the introduction of the approximate trace $\hat{u}_h$ as an explicit unknown. As a result, the computational cost and memory footprint of the hybridized DG methods is several times smaller than in other DG methods$^1$.

---

$^1$Strictly speaking, this advantage disappears if a matrix-free method is used. This is rarely the case in practice due to the difficulty to obtain a good preconditioner without explicitly forming the matrix of the system.
3.2.2 Generalized Minimal Residual method

The linear system (3.4) is solved in parallel using the restarted Generalized Minimal Residual, GMRES(\(\beta\)), method [52] with iterative classical Gram-Schmidt (ICGS) orthogonalization. The ICGS method requires less communication instances per GMRES iteration than other orthogonalization strategies and thus results in better parallel performance. Also, \(\beta\) denotes the restarting parameter and is set to \(\beta = 120\).

Parallel preconditioner: Restricted additive Schwarz method

A left preconditioner \(M^{-1} \approx K^{-1}\) is used to accelerate GMRES convergence, so that the linear system (3.4) is replaced by

\[
M^{-1} K \delta \hat{U} = M^{-1} R. \tag{3.5}
\]

In particular, we employ the restricted additive Schwarz (RAS) method [4] in order to achieve excellent scalability of the linear solver. This approach relies on a decomposition of the unknowns in \(\delta \hat{U}\) among parallel processes; which is performed as described below.

Let \(\hat{P}\) denote the set of nodes \(\{\hat{p}_1, \hat{p}_2, \ldots\}\) used to approximate the solution trace \(\hat{u}_h\), i.e., the red dots in Fig. 2-1. These nodes will be referred to as traced nodes throughout the remainder of this chapter. We first partition \(\hat{P}\) into \(N\) nonempty, nonoverlapping subdomains \(\hat{P}_i^{(0)}\); that is,

\[
\hat{P} = \bigcup_{i=1}^{N} \hat{P}_i^{(0)}, \quad \hat{P}_i^{(0)} \cap \hat{P}_j^{(0)} = \emptyset \quad \text{for } i \neq j, \quad \hat{P}_i^{(0)} \neq \emptyset \quad \text{for } i = 1, \ldots, N, \tag{3.6}
\]

where \(N\) denotes the number of physical cores. This is the “zero-overlap” decomposition and is computed by minimizing the edge-cut of the \(\hat{p}\)-to-\(\hat{p}\) connectivity graph using the METIS software [31] and a multilevel \(k\)-way partitioning algorithm [30]. The “one-overlap” decomposition is then defined as the collection of supersets of \(\hat{P}_i^{(0)}\) obtained by including all the traced nodes immediately neighboring the nodes in \(\hat{P}_i^{(0)}\). Using this idea recursively, we define a \(\delta\)-overlap decomposition \(\hat{P}_i^{(\delta)}\) for some
nonnegative integer $\delta$. Note that the $\delta$-overlap decomposition depends on which hybridized DG method is used to discretize the Navier-Stokes equations. Figure 3-1 depicts an illustration of the one-overlap decomposition for the HDG and IEDG schemes. This figure shows how the traced nodes are first decomposed into two nonoverlapping subdomains. The one-overlap decomposition is obtained by adding all the traced nodes immediately neighboring nodes in the nonoverlapping subdomains.

The RAS preconditioner is then defined by

$$M^{-1} := \sum_{i=1}^{N} R_{i}^{0} K_{i}^{-1} R_{i}^{\delta}, \quad (3.7)$$

where $K_{i} = R_{i}^{\delta} K R_{i}^{\delta}$ is the so-called subdomain problem and $R_{i}^{\delta}$ is the restriction operator onto the subspace associated to the traced nodes in the $\delta$-overlap subdomain number $i$. Note that the choice $\delta = 0$ leads to the so-called block Jacobi (BJ) preconditioner. From our experience, $\delta = 1$ provides the best balance between communication cost and number of GMRES iterations for the flow regimes considered in this work, and is therefore employed here. This is consistent with the observation in [51].
Subdomain preconditioner: Incomplete LU factorization

In practice, we replace $K_i^{-1}$ by the inverse of the block incomplete LU factorization with zero fill-in of $K_i$. We shall refer to this as the BILU(0) factorization of $K_i$. In this approach, the approximate LU factors exactly preserve the sparsity pattern of $K_i$, so that no entry in $\tilde{L}_i$ or $\tilde{U}_i$ is allowed to be nonzero outside of the sparsity pattern of $K_i$. This factorization is inexpensive to compute and requires the same memory as $K_i$. Dropping the subscript $i$ in $\tilde{L}_i$ and $\tilde{U}_i$ to simplify the notation, the BILU(0) factorization for hybridized DG methods can be computed as outlined in Algorithm 1. For the sake of simplicity, Algorithm 1 assumes that the block size is equal to the number of components of the Navier-Stokes system. A generalization for larger block sizes will be presented below.

Data: $K_i$
Result: $\tilde{L}, \tilde{U}$
$\tilde{U} \leftarrow K_i, \tilde{L} \leftarrow I$

for $\hat{p}_j$ in $\hat{P}_i^b$ do
  for $\hat{p}_l$ in $\hat{P}_i^b$, neighboring $\hat{p}_j$, and such that $l > j$ do
    $\tilde{L}_{lj} = \tilde{U}_{lj}^{-1}$
    $\tilde{U}_{ll} \leftarrow \tilde{U}_{ll} - \tilde{L}_{lj} \tilde{U}_{jl}$
    for $\hat{p}_k$ in $\hat{P}_i^b$, neighboring $\hat{p}_j$ and $\hat{p}_l$, and such that $k > j$ do
      $\tilde{U}_{lk} \leftarrow \tilde{U}_{lk} - \tilde{L}_{lj} \tilde{U}_{jk}$
    end
  end
end

Algorithm 1: BILU(0) factorization for hybridized DG methods.

Minimum Discarded Fill algorithm

The approximation provided by the BILU(0) factorization depends on the ordering of the equations in the linear system (3.5), as this determines what blocks are discarded with respect to the exact LU factors. This motivates us to compute the subdomain preconditioner in conjunction with the Minimum Discarded Fill (MDF) ordering algorithm. The MDF algorithm was introduced in [48] for standard DG methods, and its extension to hybridized DG methods here is straightforward and summarized.
in Algorithm 2. In this algorithm, $|| \cdot ||_F$ denotes the Frobenius norm, $\mathbf{M}_i^{BJ}$ is the block Jacobi approximation to $\mathbf{K}_i$, and $\mathcal{P}_i$ denotes the permutation vector to reorder the nodes in $\hat{P}_i^\delta$.

In order to improve the scalability of the RAS preconditioner, Algorithm 2 may be equipped with the constraint that the nodes in the interior of the subdomain are ordered first and the boundary nodes last. This way, the BILU(0) solve can start without the need to wait for the result of the matrix-vector product in other processors.

**Algorithm 2:** Minimum Discarded Fill (MDF) ordering for hybridized DG methods.

| Data: $\mathbf{K}_i$ |
| Result: $\mathcal{P}_i$ |
| $B \leftarrow (\mathbf{M}_i^{BJ})^{-1}\mathbf{K}_i$ |
| $C_{lj} \leftarrow ||(\mathbf{K}_i)_{lj}||_F$ |
| $D_{lj} \leftarrow ||B_{lj}||_F$ |
| for $\hat{p}_k$ in $\hat{P}_i^\delta$ do |
| $\Delta F \leftarrow 0$ |
| for $\hat{p}_l$ in $\hat{P}_i^\delta$ and neighboring $\hat{p}_k$ do |
| for $\hat{p}_j \neq \hat{p}_l$ in $\hat{P}_i^\delta$, neighboring $\hat{p}_k$, and not neighboring $\hat{p}_l$ do |
| $\Delta F_{lj} = C_{lk}D_{kj}$ |
| end |
| $\omega_k \leftarrow ||\Delta F||_F$ |
| end |
| for $l = 1, 2, ...$ do |
| $(\mathcal{P}_i)_l \leftarrow \arg\min_j \omega_j$ |
| $\omega(\mathcal{P}_i)_l \leftarrow \infty$ |
| for $\hat{p}_k$ in $\hat{P}_i^\delta$, neighboring $\hat{p}_l$, and not yet numbered do |
| Recompute $\omega_k$ only considering neighbors not yet numbered |
| end |
| end |

**Node clustering**

To simplify the discussion, we have assumed that the blocks in the BILU(0) factorization are of size $m \times m$. Next, we present a generalization of the subdomain preconditioner that operates with blocks of larger size and has important advantages in practice.
Let $\Pi_1, \Pi_2, \ldots$ be clusters of traced nodes that partition $\hat{P}_i$ (see footnote 2 for details) and such that all the nodes in each cluster satisfy the property of neighboring exactly the same traced nodes, namely, some superset of the cluster. Then, Algorithms 1-2 still apply if the role of the traced nodes $\hat{p}_j$ is replaced by that of the clusters $\Pi_j$, and then the blocks become of size $(m \cdot n_j) \times (m \cdot n_j)$, where $n_j$ denotes the cardinality of $\Pi_j$. First, operating with larger blocks improves the approximation provided by the BILU$(0)$ factorization. Second, the matrix-vector products and preconditioner solves are computed faster due to the better use of cache memory as the block size increases. We finally note that, for the HDG method, all the nodes on the same face can be grouped together, so that there is a simple one-to-one mapping between clusters and faces.

**Summary**

Our parallel preconditioner is thus given by

$$M^{-1} := \sum_{i=1}^{N} R_i^0 \tilde{U}_i^{-1} \tilde{L}_i^{-1} R_i^\delta.$$  \hfill (3.8)

Each GMRES iteration requires us to calculate $y = M^{-1}K r$, where $r$ is the current Krylov vector. This is computed in two steps as

$$x = K r, \quad y = M^{-1} x.$$  \hfill (3.9)

Next, we describe the implementation of these two operations: Matrix-vector product and preconditioner solve.

---

2That is, $\hat{P}_i = \bigcup_{j=1}^{N_c} \Pi_j$ with $\Pi_j \neq \emptyset$ for $j = 1, \ldots, N_c$, and $\Pi_j \cap \Pi_k = \emptyset$ for $j \neq k$. Here, $N_c$ denotes the number of clusters in the partition of $\hat{P}_i$. 

38
3.2.3 Implementation of the matrix-vector product and BILU(0) solve

Fast matrix-vector products and preconditioner solves are required for an efficient solution of the linear system (3.5). To that end, we exploit computer memory hierarchy and cache policies at the subdomain level. In particular, a block compressed row format is used to store $K_i$, as well as its BILU(0) factors $\tilde{L}_i$ and $\tilde{U}_i$. In order to take advantage of cache lines, the blocks are stored in memory in the same order as they will be accessed for the matrix-vector product and the preconditioner solve. For the BILU(0) factors, this is in turn determined by the MDF ordering. This storage strategy increases the hit rate and thus improves the performance of these memory bound operations. Also, the diagonal blocks of $\tilde{U}_i$ are overwritten by their LU factors when the preconditioner is formed.

The block matrix-vector products that comprise the full matrix-vector product and preconditioner solve are performed first for the nodes on the interface between subdomains and then for the interior nodes in order to overlap communication and computation as much as possible. These block matrix-vector products are computed through the Basic Linear Algebra Subprograms (BLAS). This way, we exploit the structure of $K$ and $M$ while performing the actual computations through highly optimized dense linear algebra libraries.

3.2.4 Mixed-precision algorithm

A mixed-precision algorithm is used for the GMRES iteration. In particular, the matrix-vector products and preconditioner solves are performed in single precision, while the orthogonalization is performed in double precision. Since the former are memory bound operations, a factor of two speedup is achieved. We note that the number of GMRES and Newton iterations is unaffected by the type of arithmetic. This is so because the preconditioner and the Newton linearization remain sufficiently accurate even when these operations are carried out in single precision.
3.2.5 Stopping criterion and adaptive tolerance

The stopping criterion for the GMRES iteration is based on the relative magnitude of the norm of the residual and the right-hand side in Eq. (3.5). In particular, the linear system is iterated until a quadratically convergent Newton iterate is obtained, without the need for solving the system to machine precision. This is typically achieved by reducing the initial residual by five or six orders of magnitude. Also, the GMRES tolerance is adaptively modified to avoid unnecessary iterations when the nonlinear residual is close to the nonlinear tolerance. This simple adaptive strategy can reduce the number of GMRES iterations by about 30%.
Chapter 4

Results

This chapter presents ILES results for both internal and external aerodynamic flows. Again, the discussion will be limited to the most informative results in order to highlight the key points and avoid unnecessary complexity in the exposition.

4.1 NACA 65-(18)10 compressor cascade

The proposed approach is first applied to transitional flows over a NACA 65-(18)10 linear compressor cascade at Reynolds number $Re = 250,000$ and Mach number $M_\infty = 0.081$. The blade solidity is $\sigma = 1.0$, the stagger angle $\xi = 28.3$ deg., and the freestream turbulence intensity $I_\infty = 0.0$. Two operating conditions are considered. First, the relative angle between the freestream and the blade chord is set to 16.7 deg. This corresponds to the design condition of the NACA 65-(18)10 compressor cascade at this Reynolds number. Next, the angle of attack is increased to 25.7 deg. for an analysis in off-design condition. This is 9.0 deg. larger than in design condition, and will lead to early separation on the suction side and a large laminar separation bubble on the pressure side. Our simulation results will be compared to those of a subgrid-scale LES model discretized using a second-order finite volume code and 31,000,000 elements [35].
4.1.1 Details of the numerical discretization

The IEDG method and the third-order, three-stage DIRK(3,3) scheme are used for the spatial and temporal discretization, respectively. The polynomial degree of the IEDG approximation is set to \( k = 2 \) and the non-dimensional time-step to \( dt^* = dt \cdot U_\infty / c = 0.005 \), where \( c \) denotes the airfoil chord and \( U_\infty \) is the freestream velocity. The resulting discretization is therefore third-order accurate in space and time.

The computational domain is partitioned using isoparametric hexahedral elements. In particular, the three-dimensional meshes are generated through extrusion of two-dimensional quadrilateral meshes with extrusion length equal to \( 0.1c \). The design and off-design conditions have been both simulated on a sequence of meshes until the transition location and the pressure coefficient are grid independent and thus grid converged. This leads to the four computational meshes whose details are summarized in Table 4.1. In this table, global unknowns indicates the number of unknowns in Eq. (3.5) and the \( \times 5 \) factor accounts for the five components in the Navier-Stokes system. An estimate of the computational cost on an Intel Xeon E5-2650 architecture is included as well. Figure 4-1 shows a 2D slice of Mesh No. 2.

<table>
<thead>
<tr>
<th>Mesh No.</th>
<th>( k )</th>
<th>( p )</th>
<th>No. elements</th>
<th>Element type</th>
<th>Global unknowns</th>
<th>Cores ( \times ) hours</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>2</td>
<td>95,400</td>
<td>Hexes</td>
<td>810,600 ( \times 5 )</td>
<td>64 ( \times 20.3 )</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>2</td>
<td>201,180</td>
<td>Hexes</td>
<td>1,632,828 ( \times 5 )</td>
<td>128 ( \times 21.6 )</td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>2</td>
<td>427,040</td>
<td>Hexes</td>
<td>3,365,784 ( \times 5 )</td>
<td>256 ( \times 23.0 )</td>
</tr>
<tr>
<td>4</td>
<td>2</td>
<td>2</td>
<td>909,360</td>
<td>Hexes</td>
<td>6,918,290 ( \times 5 )</td>
<td>512 ( \times 24.5 )</td>
</tr>
</tbody>
</table>

Table 4.1: Details of the computational meshes considered for the NACA 65-(18)10 compressor cascade.

4.1.2 Grid convergence study and comparison to second-order SGS-LES

Figure 4-2 (left) shows the time and spanwise average of the negative pressure coefficient in design condition computed with the high-order ILES and the second-order FV SGS-LES [35]. The ILES and the SGS-LES results show very good agreement, except
Figure 4-1: 2D slice of Mesh No. 2 for the NACA 65-(18)10 compressor cascade.

on the transition location on the pressure side. This in turn induces a discrepancy on
the reattachment location and the subsequent pressure rise, as illustrated in Fig.
4-2. Similarly, Fig. 4-3 shows the negative of the spanwise- and time-averaged
pressure coefficient in off-design condition. The disagreement on the transition lo-
cation between the high-order ILES and the second-order SGS-LES is more profound
than in design condition. Since the transition location and the pressure coefficient
seem grid converged in the high-order ILES results for both operating conditions,
these discrepancies can be attributed to (1) unphysical dissipation introduced by the
SGS model in the laminar portion of the boundary layer, (2) under-resolution in the
FV SGS-LES results, or (3) other numerical artifacts in either simulation, such as
numerical dissipation, numerical dispersion, and spurious reflections on the inflow
and outflow boundaries of the computational domain.

We emphasize that high-order ILES achieves grid convergence on the transition
location and the pressure coefficient with two orders of magnitude fewer elements
and one order of magnitude fewer degrees of freedom\footnote{We use the term “degrees of freedom” here as an indicator of the resolution provided by our high-order DG discretization and enable comparison with low-order FV methods. For example, a “5M DOFs” high-order solution has the same number of effective grid points as a cell-centered finite volume code with 5M elements.} (DOFs) than those used with
the second-order FV SGS-LES. While few grid convergence studies are available in
the literature of LES for turbomachinery flows, the spatial resolution required by the
high-order ILES is well below the ones typically used with low-order LES approaches.
An explanation for high-order ILES being able to accurately capture transition with such few elements and degrees of freedom will be presented in Chapter 5.

### 4.1.3 Boundary layer analysis

We investigate the structure of the boundary layer (BL) and the transition mechanism using the high-order ILES results. The BL analysis is presented first for the design condition.

**Figure 4-2:** Negative spanwise- and time-averaged pressure coefficient computed with high-order ILES and second-order FV SGS-LES for NACA 65-(18)10 in design condition. Iso-surface of the Q-criterion [26] colored by pressure is shown on the right.

**Figure 4-3:** Negative spanwise- and time-averaged pressure coefficient computed with high-order ILES and second-order FV SGS-LES for NACA 65-(18)10 in off-design condition. Iso-surface of the Q-criterion [26] colored by pressure is shown on the right.
condition and then for the off-design condition. The nomenclature and details of the post-processing strategy are described in Appendix A.

The flow separates on the suction side at $x_s/c = 0.50$ in design condition due to the adverse pressure gradient (Fig. 4-2). This produces a laminar separation bubble (LSB) and strongly destabilizes the boundary layer; which eventually transitions to turbulence. After transition, the turbulent mixing leads to rapid reattachment at $x_r/c = 0.71$ and the separation bubble ends. The turbulent BL then remains attached all the way until the trailing edge thanks to the resistance to separation provided by the turbulent mixing. A similar behavior is observed on the pressure side, with BL separation also at $x_s/c = 0.50$ and reattachment at $x_r/c = 0.71$. The laminar separation and turbulent reattachment are illustrated in Fig. 4-4 through the instantaneous and time-averaged velocity magnitude fields. Similarly, the LSBs translate into the pressure plateaux in Fig. 4-2, followed by a rapid pressure rise after transition. These plateaux are a consequence of the nearly-still fluid inside the laminar part of the bubble being unable to sustain any significant pressure gradients. The turbulent mixing in the turbulent portion of the bubble, however, can support the strong adverse pressure gradient in Fig. 4-2 [8].

![Figure 4-4: Instantaneous (left) and time-averaged (right) velocity magnitude fields for NACA 65-(18)10 in design condition.](image)

Also, separation leads to an increase in the displacement thickness and thus the shape parameter, as illustrated in Fig. 4-5 for the suction side. The increase in the shape parameter will be (indirectly) responsible for the rapid growth of instabilities along the bubble, as discussed below. The large value of the displacement thickness
along the adverse pressure gradient in the turbulent portion of the bubble is responsible for the so-called bubble drag and produces additional losses in the cascade. Once the LSB ends, the displacement thickness and the shape parameter reduce back to attached regime values. Then, the strong pressure gradient from the reattachment location to the trailing edge leads to the final increase in the displacement and momentum thickness in Fig. 4-5.
Due to the lack of bypass and forced transition mechanisms and the quasi-2D nature of the flow, natural transition through two-dimensional unstable modes is expected. This is numerically confirmed by the high-order ILES results. The two-dimensional nature of transition is illustrated in Fig. 4-6 through the much larger amplitude of the streamwise instabilities compared to the cross-flow instabilities. In particular, Tollmien-Schlichting (TS) waves form before the boundary layer separates, and Kelvin-Helmholtz (KH) instabilities are ultimately responsible for transition after separation\(^2\). The former are shown in Fig. 4-7 (left) through the Gaussian-like shape of the root mean square (RMS) in time of the fluctuating velocity magnitude at different BL locations prior to separation. More specifically, the left plot in Fig. 4-7 shows the superposition of (1) TS waves and (2) pressure waves generated in the turbulent boundary layer of the blade at hand and the neighboring blade. The latter effect is responsible for the nonzero fluctuating velocity outside the boundary layer.

\[\text{Figure 4-7: TS waves (left) and streamwise amplification factor (right) on the suction side for NACA 65-(18)10 in design condition.}\]

The growth rate of TS waves along the BL is exponential, as shown on the right of Fig. 4-7 and predicted by linear stability theory. The box on the right figure indicates the region of the BL in which the TS waves on the left are located. It is worth

\(^2\)We consider TS and KH waves to be different phenomena. In particular, we refer to the unstable modes of the Orr-Sommerfeld equation as TS modes if the boundary layer is attached, and as KH modes if the boundary layer is separated.
noting the small magnitude of the instabilities compared to the freestream velocity. This shows why very small amount of numerical dissipation is required for transition prediction. Similarly, very low numerical dispersion is needed to properly resolve all the frequencies present in the transition process. After separation, TS waves turn into KH instabilities as illustrated in Fig. 4-8. KH modes produce very rapid vortex growth and are ultimately responsible for natural transition in the separated shear layer. Turbulence constitutes a new, stable flow regime that prevents instabilities to keep growing exponentially in space and time.

Figure 4-8: Transition from TS to KH modes along the separated, suction side boundary layer for NACA 65-(18)10 in design (left) and off-design (right) conditions.

The non-dimensional velocity profile at different locations along the turbulent portion of the suction side BL are displayed in Fig. 4-9 for the inner (left) and outer (right) layers. The high-order ILES properly captures the viscous sublayer $u^+ = n^+$, which extends from $n^+ = 0$ to $n^+ \approx 8$. Also, a log-layer $u^+ = (1/\kappa) \log (n^+) + C^+$ is observed from $n^+ \approx 20$ to $n^+ \approx 200$, where the exact extremes depend on the local Reynolds number and the pressure gradient. For moderate pressure gradients, the numerical results fit well the experimentally measured value for the von Kármán constant $\kappa \approx 0.40$ (and $C^+ \approx 5.5$), while smaller values of $\kappa$ are predicted for strong adverse pressure gradients.

Next, we discuss the structure of the BL and the transition mechanism in off-
Figure 4-9: Non-dimensional turbulent velocity profiles in the inner (left) and outer (right) layers at different locations along the suction side turbulent BL for NACA 65-(18)10 in design condition.

Figure 4-10: Instantaneous (left) and time-averaged (right) velocity magnitude fields for NACA 65-(18)10 in off-design condition.

design condition. Some details will be omitted in this case due to the similarities with the design condition. The boundary layer again undergoes laminar separation and turbulent reattachment on both the suction and pressure sides, hence leading to a LSB. The separation and reattachment locations are $x_s/c = 0.20$ and $x_r/c = 0.38$ on the suction side, and $x_s/c = 0.57$ and $x_r/c = 0.89$ on the pressure side. This is illustrated in Fig. 4-10 through the instantaneous (left) and time-averaged (right) velocity magnitude fields. The existence of separation bubbles is also clear from the pressure plateaux in Fig. 4-3, as well as from the streamwise displacement thickness and shape parameter in Fig. 4-11. It is worth noting the large size of
Figure 4-11: Streamwise displacement and momentum thickness (left) and shape parameter (right) along the pressure side for NACA 65-(18)10 in off-design condition.

Figure 4-12: Amplitude of streamwise and cross-flow instabilities (left) and TS waves (right) on the suction side for NACA 65-(10)18 in off-design condition.

the separation bubble on the pressure side; which is typical of off-design operating conditions at low Reynolds numbers. Also, the larger angle of attack with respect to the design condition moves the LSB downstream on the pressure side and upstream on the suction side. In particular, early separation and transition allow for a more gradual pressure recovery along the suction side, and this translates into a suboptimal tangential force on the blade. Again, TS waves and KH instabilities are responsible for natural transition to turbulence, as illustrated in Figure 4-12 for the suction side.
Finally, Figure 4-13 shows the non-dimensional velocity profiles at different locations along the suction side turbulent BL for the inner (left) and outer (right) layers.

![Figure 4-13: Non-dimensional turbulent velocity profiles in the inner (left) and outer (right) layers at different locations along the suction side turbulent BL for NACA 65-(18)10 in off-design condition.](image)

### 4.2 Eppler 387 wing

The results for the NACA 65-(18)10 compressor cascade demonstrate the rapid grid convergence of high-order ILES to capture transition. Next, we examine the accuracy of the high-order ILES transition prediction capability by comparing it to experimental data. In this spirit, we consider transitional flows over the Eppler 387 wing at Reynolds numbers of 100,000, 300,000, and 460,000. The angle of attack is set to $\alpha = 4.0$ deg. and the freestream turbulence intensity to $I_\infty = 0.0$. The Mach number $M_\infty$ is 0.08, 0.09 and 0.13 for the three Reynolds numbers considered, respectively. As it happens, the experimental data for the Eppler wing [34] are arguably the most accurate and reliable in the literature of transitional flows. This justifies the choice of this test case for validation purposes.
4.2.1 Details of the numerical discretization

The HDG method and the DIRK(3,3) scheme are used for the spatial and temporal discretization of the Navier-Stokes equations, respectively. The polynomial degree of the numerical approximation is \( k = 4 \), hence yielding a scheme that is fifth-order in space and third-order in time. The computational domain is partitioned using isoparametric tetrahedral elements. The extrusion length in the spanwise direction is \( 0.1c \) and the computational domain extends about 10 chords away from the wing in the radial direction. Three meshes and non-dimensional time-steps are considered; which correspond to uniform refinement in space and time. The details of these meshes are summarized in Table 4.2. Again, global unknowns indicates the number of unknowns in Eq. (3.5) and the ×5 factor accounts for the five components in the Navier-Stokes system. Figure 4-14 shows a 2D slice of Mesh No. 2.

<table>
<thead>
<tr>
<th>Mesh No.</th>
<th>( k )</th>
<th>( p )</th>
<th>No. Elements</th>
<th>Element type</th>
<th>Global unknowns</th>
<th>( dt^* )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4</td>
<td>4</td>
<td>64,800</td>
<td>Tets</td>
<td>1,959,600 × 5</td>
<td>7.937E−3</td>
</tr>
<tr>
<td>2</td>
<td>4</td>
<td>4</td>
<td>126,360</td>
<td>Tets</td>
<td>3,814,380 × 5</td>
<td>6.300E−3</td>
</tr>
<tr>
<td>3</td>
<td>4</td>
<td>4</td>
<td>254,976</td>
<td>Tets</td>
<td>7,687,680 × 5</td>
<td>5.000E−3</td>
</tr>
</tbody>
</table>

Table 4.2: Details of the computational meshes considered for the Eppler 387 wing.

Figure 4-14: 2D slice of Mesh No. 2 for the Eppler 387 wing. Tetrahedral meshes are created by subdividing each hexahedron into 6 tetrahedra.
4.2.2 Grid convergence study and comparison with experimental data

The negative spanwise- and time-averaged pressure coefficient at Reynolds numbers 100,000, 300,000 and 460,000 are shown in Figures 4-15, 4-16 and 4-17, respectively. The ILES simulations agree very well with the experimental data reported in [34]. Furthermore, we observe grid convergence in the sense that the simulation results
collapse as the mesh is refined. More importantly, the error in the transition location is below 0.01c, 0.005c and 0.01c at Reynolds number 100,000, 300,000 and 460,000, respectively, even with Mesh No. 1. The effective resolution of this mesh is equivalent to a cell-centered finite volume discretization with 691,200 elements. Again, these numbers are much below those needed with a low-order SGS-LES approach. Also, the transition from laminar to turbulent regime can be more accurately predicted at Reynolds number of 300,000 than 100,000. As discussed in the next section, this is due to the differences in the transition process between both Reynolds numbers.

![Fig. 4-17: ILES prediction of the pressure coefficient (left) and skin friction coefficient (right) for the Eppler 387 wing at \( Re = 460,000 \).](image)

The missing vortex upwash due to the finite extent of the computational domain leads to a change in the effective angle of attack. This affects the pressure field around the wing and is responsible for the mismatch near the leading edge with respect to the experimental data. This motivates us to extend the computational domain 50 chords away from the wing, and compare the results to those with a 10-chord domain and the experimental data. In particular, the 50-chord mesh is an extension of the 10-chord Mesh No. 3 following the same geometric growth rate for the element size along the radial direction as in the first 10 chords. The results at Reynolds number 100,000 are shown in Fig. 4-18. The change in the effective angle of attack is inversely proportional to the radius of the computational domain and, in particular, a 50-chord domain leads to \( \Delta \alpha = 0.0709 \) deg. Also, the wind tunnel walls in [34] were located 13
chords away from the wing; which makes the experimental data susceptible to small inaccuracies as well. Overall, the agreement between the experimental data and the ILES results is very good, and the minor discrepancies are within expected due to experimental errors, the nonzero freestream turbulence level in the experimental setting, the finite extent of the physical and computational domains, etc.

Figure 4-18: Pressure coefficient near the leading edge for different sizes of the computational domain for the Eppler 387 wing at $Re = 100,000$.

4.2.3 Boundary layer analysis

The boundary layer remains attached along the pressure side for the three Reynolds numbers considered. In particular, the design of the Eppler 387 wing is such that the adverse pressure gradient on the lower side takes place only in a very small region near the leading edge, where the momentum thickness is small and the boundary layer therefore very resistant to separation. The greater stability of the attached BL translates into a fully laminar flow along the pressure side despite the relatively large value of the Reynolds number. In fact, the amplitude of the TS waves remains at least two orders of magnitude below the freestream velocity, as shown on the right of Figures 4-19 and 4-20.
Figure 4-19: Amplitude of streamwise and cross-flow instabilities along the suction (left) and pressure (right) sides for the Eppler 387 wing at Reynolds number 100,000.

The suction side boundary layer, however, does undergo laminar separation and turbulent reattachment for the three Reynolds numbers. In particular, the separation bubble extends from $x_s/c = 0.38$ to $x_r/c = 0.80$ at $Re = 100,000$, from $x_s/c = 0.46$ to $x_r/c = 0.55$ at $Re = 300,000$, and from $x_s/c = 0.46$ to $x_r/c = 0.52$ at $Re = 460,000$. This is reflected in the pressure plateaux in Figures 4-15, 4-16 and 4-17, the average velocity magnitude fields in Fig. 4-21, and the increase in the displacement.
thickness and the shape parameter in Figures 4-22 and 4-23. Also, it results in rapid amplification of KH instabilities in the separated shear layer and eventual transition to turbulence, as shown on the left of Figures 4-19 and 4-20. Since the process is analogous to that described for the NACA 65-(18)10 compressor blade, the details are omitted here.

At Reynolds numbers of 300,000 and 460,000, the amplitude of the BL instabilities at the separation location is only one order of magnitude below the transition threshold, hence leading to rapid transition and reattachment. The small size of the LSB has important consequences from a physical and numerical point of view. From a physical perspective, it reduces the bubble drag and thus the overall drag coefficient. From a numerical perspective, it makes transition easier to predict. This is in turn due to
Figure 4-22: Streamwise displacement and momentum thickness (left) and shape parameter (right) along the suction (top) and pressure (bottom) sides for the Eppler 387 wing at Reynolds number 100,000.

the transition process taking place under smaller values of the shape parameter $H$. Indeed, although the length and time scales of the BL instabilities scale as $Re^{-1/2}$ for a fixed value of $H$, this may be leveraged by the smaller value of the shape parameter itself and translate into the most unstable BL modes having larger length and time scales. This is the case at $Re = 300,000$ vs. 100,000, and demonstrates that the transition mechanism plays a key role in the resolution needed to capture transition and that resolution estimates solely based on dimensional analysis are likely to fail for the transition problem.

We finally note that the separation location coincides at Reynolds numbers of

58
300,000 and 460,000. This is consistent with the small separation bubble having a negligible effect on the potential flow field around the wing, as illustrated in Fig. 4-21, and the Reynolds number independenec of laminar boundary layer flows [8]. As such, the displacement and momentum thickness (upon scaling by $Re^{1/2}$) and the shape parameter at $Re = 460,000$ look indistinguishable to those at 300,000, and for this reason have been omitted here.

Figure 4-23: Streamwise displacement and momentum thickness (left) and shape parameter (right) along the suction side for the Eppler 387 wing at Reynolds number 300,000.
Chapter 5

Rationale for high-order ILES for transition prediction

Whether high-order ILES can provide any advantages to the computational fluid dynamics community remains an open question, mostly due to the limited number of efforts in this direction. While numerical results in Chapter 4 demonstrated the potential of this approach for the simulation of transitional turbulent flows, a justification for this empirical observation is still missing. In this spirit, this chapter aims to present a rationale for the success of high-order ILES for transition prediction.

5.1 Rationale for high-order methods

Two main advantages could be argued in favor of high-order methods for transition prediction. First, they may be able to capture transition in situations in which low-order schemes fail to give accurate results. Second, for some intended accuracy, they could provide a numerical solution in less computational time. These two points are actually related. To the author’s opinion, low-order schemes can always be accurate enough if the spatial and temporal resolutions are sufficiently fine. However, this may render them so computationally expensive that they become less competitive than their high-order counterparts. As discussed in previous chapters, transition is a very subtle process and high-order accuracy is key to capture the delicate physical
phenomena that take place during transition from laminar to turbulent regime.

Indeed, our numerical results showed that high-order methods require significantly less elements and degrees of freedom than low-order methods by one or two orders of magnitude. This is justified by the following observation: *Simulating transition is challenging mostly due to the small magnitude of the instabilities involved, rather than due to their length and time scales.* A low-order scheme might kill the small instabilities because of high numerical dissipation even when the mesh size and time step are sufficiently small to represent the length and time scales of the instabilities. The number of elements and degrees of freedom to capture transition can therefore be significantly reduced by increasing the accuracy order of the numerical scheme.

Whether this reduction in elements and degrees of freedom translates into lower computational cost is a separate, and ultimately more important, question. In particular, while our high-order approach is currently about the same cost as a second-order finite volume LES code, we anticipate an advantage for high-order methods in the future. First, low-order codes are limited by memory bandwidth and thus doomed to perform well below the peak performance of the processor. The high operational intensity of high-order methods, however, allows them to get much closer to the peak performance. This is key for the upcoming high-performance computing (HPC) paradigm. Second, high-order, implicit-in-time codes are dominated by the solution of the nonlinear system of equations and therefore have much room for improvement as new solution methods are developed.

Furthermore, for the flow regimes in which the transition location plays a key role in the aerodynamic performance, the element size near transition is usually much larger than near the leading edge. Hence, the temporal scales of the large eddies in the turbulent portion of the boundary layer can be accurately represented with Courant numbers greater than $\mathcal{O}(1)$. This can be exploited by using an $L$-stable (and thus implicit) time integration scheme and can be considered as yet another advantage of our high-order approach for transition prediction.
5.2 Rationale for ILES

The ILES approach for transition prediction can be justified by the following observation: *While some dissipation is required to account for the unresolved scales in a turbulent flow, this is not the case along the laminar portion of the boundary layer in which the transition process takes place.* As such, directly removing spurious diffusion coming from a SGS model, as ILES does, turns out to be a better approach. Whether ILES is also preferred for the turbulent portion of the boundary layer is the subject of ongoing research. In any event, the variational formulation in high-order DG methods provides an excellent framework to pursue other approaches, including dynamic explicit LES [20] and the Variational Multiscale (VMS) method [7,23–25,38].
Chapter 6

Conclusions and future work

6.1 Conclusions

We presented a high-order Implicit Large-Eddy Simulation (ILES) approach to simulate transitional turbulent flows. In particular, this thesis contains novel contributions in terms of methodology and applications. From the methodology standpoint, we proposed using hybridized Discontinuous Galerkin (DG) methods [44] for Large-Eddy Simulation, and developed a parallel preconditioned Newton-GMRES solver for the nonlinear system of equations arising from the numerical discretization. Regarding the nonlinear solver, we introduced a minimal residual algorithm to compute a good initial guess for the Newton iteration. We also described an overlapping domain decomposition and applied it to construct a restrictive additive Schwarz parallel preconditioner. A block incomplete LU factorization with zero fill-in and the Minimum Discarded Fill reorder algorithm [48] were extended to hybridized DG methods and employed for the subdomain preconditioner. The combination of hybridized DG methods with an efficient solution procedure resulted in a high-order accurate Navier-Stokes solver that is competitive with low-order codes in terms of computational cost.

From the applications perspective, this thesis is, together with preliminary results in [10, 11], the first attempt at high-order DG ILES at Reynolds numbers up to 460,000 that is far beyond the existing simulations in the literature. In particular, the high-order ILES approach was applied to transitional flows over a NACA 65-
10 compressor cascade at Reynolds number of 250,000 and the Eppler 387 wing at Reynolds numbers of 100,000, 300,000 and 460,000. Numerical results showed rapid grid convergence and excellent agreement with experimental data. The number of degrees of freedom to accurately capture transition was found to be smaller than in low-order methods by one order of magnitude. A theoretical justification for the success of high-order ILES for transition prediction was presented. Emphasis was also placed on analyzing the structure of the boundary layer and the transition mechanism. Tollmien-Schlichting and Kevin-Helmholtz instabilities were numerically detected and concluded to be responsible for natural transition to turbulence through a laminar separation bubble.

Finally, it is known that high-order methods become less and less efficient as the order of accuracy increases beyond a certain value. Hence, there will be an optimal range for the order of accuracy to yield the best trade-off between accuracy and efficiency. Through the author’s experience, the hybridized DG methods reach their best performance when the polynomial degree $k$ is in the range between 2 and 4; that is, when the order of accuracy is between 3 and 5. Again, numerical results presented in this thesis indicate that this range is adequate to accurately and efficiently resolve the transition process at Reynolds number up to 460,000.

### 6.2 Future work

This thesis constitutes a preliminary effort in the analysis and application of high-order ILES to computational aerodynamics and, more generally, computational physics. As such, future work is only limited by imagination. The topics that, to the author’s opinion, are most important due to their relevance and potential impact are summarized below.

From the point of view of methodology, a number of strategies to further reduce the computational cost could be considered. First, new methods for the solution of the nonlinear system of equations could be investigated. In this direction, we propose exploiting the convex nature of the system in the limit of small time steps.
In particular, the use of first-order methods from the optimization community seems a promising approach for Large-Eddy Simulation. Also, the development of improved preconditioners for the Navier-Stokes equations\(^1\) has great potential to bring down the computational cost.

From an applications perspective, the extension to separated and fully turbulent flows at high Reynolds numbers is the next logical step. In addition to ILES, a variety of approaches to account for the unresolved scales of the turbulent flow could be investigated, including explicit LES and the Variational Multiscale (VMS) method [7, 23–25, 38]. Also, the approach could be extended to transonic, supersonic and hypersonic flows; which would in turn require the development of a robust shock capturing capability. To that end, we suggest borrowing ideas from the VMS framework to improve stability in the presence of high-order modes in the test space. Finally, multiphysics applications such as chemistry (combustion) and plasma physics (lightning strike) could benefit from high-order ILES and therefore be considered.

---

\(^1\)In the context of first-order methods, a preconditioner can be thought of as a *discrete dynamics accelerator near the root*. 
Appendix A

Boundary layer post-processing

This Appendix introduces the nomenclature and methodology employed for the boundary layer (BL) post-processing presented in Chapter 4. The BL analysis is based on a pseudo-velocity defined as

\[ u^* (s, n) := \int_0^n (\omega \times \hat{n}) \, dn', \]  

(A.1)

where \( \omega \) denotes the vorticity, and \((s, n)\) is the set of curvilinear coordinates associated to the airfoil surface. In particular, \( s = (s_1, s_2) \) and \( n \) are the coordinates along the streamwise, cross-flow, and outward normal to the airfoil directions, respectively. Also, the unit vectors associated to these coordinates are denoted by \( \hat{s}_1, \hat{s}_2 \) and \( \hat{n} \), respectively. Unlike the actual velocity, this pseudo-velocity asymptotes to a constant outside the boundary layer, even with strong curvature, thus making the edge of the boundary layer a well-defined location.

In particular, the BL edge \( n_e \) is computed as the first location along \( n \)-direction simultaneously satisfying

\[ \| \bar{\omega} \| \, n < \epsilon_1 \| \bar{u}^* \|, \quad \| \frac{\partial \bar{\omega}}{\partial n} \| \, n^2 < \epsilon_2 \| \bar{u}^* \|, \]  

(A.2)

where \( \epsilon_1 = 0.01 \) and \( \epsilon_2 = 0.1 \) are some properly tuned constants for a systematic and robust detection of the BL edge, and the overbar denotes temporal and cross-flow
averaging, i.e.,

\[
\bar{\omega}(s_1, n) := \frac{1}{T \cdot \Delta s_2} \int_0^T \int_0^{\Delta s_2} \omega(s, t) \, ds_2 \, dt.
\]  

(A.3)

Cross-flow averaging corresponds to ensemble averaging due to the quasi-2D nature of the geometry and boundary conditions, and is used here to accelerate the convergence of the statistics of the turbulent flow.

The local streamwise and cross-flow unit vectors are defined as

\[
\hat{s}_1(s_1) := \frac{\bar{u}_e}{\bar{u}_e}, \quad \hat{s}_2(s_1) := \hat{s}_1 \times \hat{n},
\]  

(A.4)

where \( u_e = u^*(n_e) \) is the pseudo-velocity at the edge of the boundary layer and \( u_e = ||u_e||_2 \) is its magnitude. Also, the average \( \bar{u}_1 \) and fluctuating \( u'_1 \) streamwise velocity are given by

\[
\bar{u}_1(s_1, n) = u^*(s_1, n) \cdot \hat{s}_1(s_1, n), \quad u'_1(s, t) := u_1(s, t) - \bar{u}_1(s_1, n),
\]  

(A.5)

whereas the streamwise displacement thickness, momentum thickness and shape parameter read as

\[
\delta^*(s_1) := \int_0^{n_e} \left( 1 - \frac{\bar{u}_1}{u_e} \right) \, dn, \\
\theta(s_1) := \int_0^{n_e} \left( 1 - \frac{\bar{u}_1}{u_e} \right) \frac{\bar{u}_1}{u_e} \, dn, \\
H(s_1) := \frac{\delta^*}{\theta}.
\]  

(A.6)

The amplitude \( A_1 \) and amplification factor \( N_1 \) of streamwise perturbations at the boundary layer location \( s_1 \) are given by

\[
A_1(s_1) = \frac{1}{u_e(s_1) \sqrt{n_e(s_1)}} \sqrt{\int_0^{n_e} u'^2_1 \, dn}, \quad N_1(s_1) := \ln \left( \frac{A_1(s_1)}{A_{1,0}} \right),
\]  

(A.7)

where \( A_{1,0} \) is some reference amplitude. We note that \( A_{1,0} \) shifts \( N_1(s_1) \) by a constant factor but it does not affect its growth rate. The cross-flow version of the previous quantities are defined in an analogous manner.
Finally, the non-dimensional velocity $u^+$ and distance to the wall $n^+$ in the inner layer of the turbulent BL read

$$u^+ := \frac{\bar{u}_1}{u_\tau}, \quad n^+ := \frac{n}{l_\tau},$$  \hspace{1cm} (A.8)

where $u_\tau = \sqrt{\tau_w/\rho}$ denotes the shear velocity and $l_\tau = \nu/u_\tau$ is the wall unit length. Similarly, the non-dimensional velocity defect $\Delta u^+$ and distance to the wall $\eta$ in the outer layer are given by

$$\Delta u^+ := \frac{\bar{u}_1 - u_e}{u_\tau}, \quad \eta := \frac{n}{\delta^*}.$$  \hspace{1cm} (A.9)

All these BL quantities are necessary to analyze and understand the structure of the boundary layer and the transition mechanism, as discussed in Chapter 4.
Bibliography


