## Forward-Facing Steps Induced Transition in a Subsonic Boundary Layer

Forward-facing steps (FFS) are common configurations in engineering applications. For examples, minor errors in installation of tile-shaped surfaces and wind turbines located on hills can both be categorized into forward-facing step flows. Despite its simple two-dimensional geometry, the flow over a forward-facing step can be quite complicated. In this paper, forward-facing steps immersed in a subsonic boundary layer are surveyed through a high-order Computational Fluid Dynamics (CFD) method, in order to investigate the flow transition phenomena induced by the forward-facing step.

## Introduction

The flow phenomenon of transition from laminar to turbulent flow has long attracted the attention of researchers. On one hand, it is one of the problems that is not totally solved in classic Newtonian mechanics, while on the other hand, it is commonly observed in the engineering field as well. However, the intrinsic nonlinearity of the Navier-Stokes equation adds to the difficulty of understanding the transition phenomena.

The nonlinearity property also gives rise to some complicated flow physics. For example, for two-dimensional geometries such as Forward-Facing Steps (FFS) and Backward-Facing Steps (BFS), a two-dimensional inflow condition may induce three-dimensional flow structures, which is a key step in transition from laminar to turbulent flows.

Although similar in geometry configuration, BFS has accumulated much more detailed research results than FFS. For instance, Brederode and Bradshaw [1] studied the separation region of BFS experimentally, and Eaton and Johnston [2] reviewed the reattachment in the downstream of BFS. Nevertheless, there is little research on FFS. XXX

The papers referred above mainly falls into two Reynolds number ranges. One is the low-Reynolds-number case, such that the inlet flow is laminar, and the evolution of small disturbances is analyzed. The other is the high-Reynolds-number case, where the upstream flow is fully turbulent, and the basic focus is pressure distribution and flow regime. However, few research has been published for the Reynolds number in between, which is called the mid-Reynolds-number case here. In this range, the inlet flow is laminar, yet the FFS may induce flow transition.

Edelmann [] focused on the transitional effects of the FFS in boundary layers. A blowing-suction pulse disturbance is set on the wall upstream of the FFS, and the evolution of this wave package is studied. Numerous different flow cases are considered, including a Mach number range from 0.15 to 1.06, Reynolds number based on the step height from 400 to 4440, and zero-or-favorable streamwise pressure condition. Results were obtained by linearized flow instability analysis, and direct numerical simulation of the disturbance Navier-Stokes equation.

It is well known that there are two types of transition in boundary layer flows, that is, natural transition and bypass transition. In a natural transition, the inflow disturbances, typically the Tollmien-Schlichting waves, experience an exponential growth in the first stage, then the nonlinearity brings up disturbance in different wavelengths, and finally the vortices break down, where the flow becomes fully turbulent. By contrast, the linear stage does not exist in the bypass transition, due to the larger amplitude of inflow disturbance, or other kinds of disturbances, such as obstacles on walls and fluctuations from outside of the boundary layer.

Previous studies have demonstrated that three-dimensional obstacles in boundary layers could trigger flow transition for flows at a certain Reynolds number range. XXX. However, how a two-dimensional obstacle, such as the FFS, could affect the transitional boundary layer is not clear yet. The key difference here is that, three-dimensional horse-shoe vortices which plays a key role in obstacles flows does not exist in the FFS flow, therefore the mechanism in boundary-layer transition should be distinct.

This paper presents a numerical study of the influence of FFS on the transition of boundary-layer flows, based on the high-order Flux Reconstruction method with the Implicit Large Eddy Simulation approach. In Section 2, a brief introduction of the numerical methods is given. Then in Section 3, the cases are described, and the flow physics are studied in detail. Finally the conclusions are drawn in Section 4.

## Numerical Methods

Transitional flows are governed by compressible Navier-Stokes equations:

XXX

Such flows are dominated by waves and vortices of different sizes. Thus, a numerical method that can resolve a wide range of length scales is required. Among traditional CFD methods, the Finite Difference Method (FDM) is usually applied to such flows, due to its high resolution ability. However, boundary conditions are not easy to treat in FDM, since the scheme is not compact.

On the other hand, the resolution ability of high-order Finite Element Methods (FEM), such as Flux Reconstruction (FR) and Discontinuous Galerkin (DG), is similar to that of FDM, meanwhile the numerical scheme involves one layer of neighboring cells. Therefore, it is much easier to apply boundary conditions in high-order FEM. In transitional flows, both solid wall and outflow boundaries could play important roles in numerical simulations, and the advantages of high-order FEM are not trivial.

In this paper, an in-house CFD solver called MUltiphysics SImulation Code (MUSIC) is adopted to calculate the transitional flows induced by forward-facing steps. MUSIC is a high-order Flux Reconstruction solver on unstructured meshes. Here a brief introduction of the numerical method is demonstrated.

### Basic Ideas of the Flux Reconstruction method

The Flux Reconstruction method was first introduced by Huynh in hexahedral cells [], and later Wang [] developed this method to other kinds of cells including simplexes. The FR method can be categorized into discontinuous finite element methods. It involves multiple degrees of freedom (DOFs) in one computation cell, just like standard FEM, and uses Riemann solvers to calculate fluxes on cell interfaces, which resembles Finite Volume Method (FVM).

The DOFs in cells are variables at certain points called Solution Points (SP), and flux between cells are evaluated at points on cell interfaces called Flux Points (FP). In order to achieve the best numerical accuracy, Gauss quadrature points are chosen as SPs and FPs.

The first step of the FR method is to reconstruct a polynomial of flux in a cell based on variables at SPs. This polynomial is discontinuous at cell interfaces, thus not applicable directly. Nevertheless, a Riemann solver can be employed to get a common flux at FPs, and a correction to the flux polynomial is added. Then, the derivative of the corrected polynomial is used to propagate to the next time step.

These two steps are known as the reconstruction step and the correction step respectively. Therefore, the Flux Reconstruction method is called Correction Procedure via Reconstruction as well.

### Calculation of the Viscous Term

The procedure described above is only available for the first-order convection term, and the second-order viscous term should be calculated in a different way. Here, an additional set of equations is introduced to evaluate the gradient of conservative variables:



This augmented equation is also a first-order partial differential equation (PDE), and can also be solved by the FR method, although the cost may be high. Bassi and Rebay [] developed a simple algebraic scheme called BR2 to achieve the solution approximately, and Huynh [] introduced this scheme to the FR method. The BR2 scheme is a compact one involving only the immediate neighboring cell, and also purely symmetric, which mimics the elliptic property of the viscous term.

### Time Integration

Since an Implicit Large Eddy Simulation (ILES) approach is to be used, the calculation has to take the unsteady effect into account. For the high-order Flux Reconstruction method, explicit Runge-Kutta time integration could be an appropriate choice, because it finds a balance between precision and efficiency. Here, a Strong Stability Preserving (SSP) third-order Runge-Kutta [] scheme is adopted. The coefficients in the SSP Runge-Kutta scheme are carefully tuned to improve the numerical stability, and the maximum CFL number could reach 1.0. The scheme is as follows:



## Results and Discussion

### Case Description

XXX

The height of FFS (h) is one-third of the thickness of boundary layer at inlet plane, and the spanwise range is 30 times of h.

### Flow Physics

1. FFS vs flat plate
2. Study of numerical accuracy

## Conclusion

A transitional flow over a Forward Facing Step is studied by Implicit Large Eddy Simulation with a high-order Flux Reconstruction Method. Comparison between a flat plate and FFS shows that the FFS plays a key role in inducing flow transition.

[1] Brederode V, Bradshaw P (1972) Three-dimensional flow in nominally two-dimensional separation bubbles. I. Flow behind a rearward-facing step. Imp Coll Aero Rep 72–19

[2] Eaton JK, Johnston JP (1981) A review of research on subsonic turbulent flow reattachment. AIAA J 19:1093–1100