EVALUATION OF IMPLICIT LES MODELING OF SEPARATED FLOWS IN A BACKWARD-FACING STEP

Fatemeh Malmir¹, Jérôme Vétel¹∗

¹Department of Mechanical Engineering, Polytechnique Montréal, Montréal, Québec, Canada
*jerome.vetel@polymtl.ca

Abstract—This paper assesses the implicit Large Eddy Simulation (iLES) technique to model a separated flow over a backward-facing step (BFS), in comparison with results achieved by Direct Numerical Simulation (DNS). The iLES technique, implemented in the open-source code "Incompact3d", relies on introducing an artificial viscosity in the discretization of the viscous term to control spurious oscillations. To the best of the author’s knowledge, this method has not been tested on wall-bounded separated flows where modeling wall regions with classical LES techniques is challenging. An internal flow over a BFS is simulated at Reynolds number Re = 5000 and expansion ratio Er = 2. Two inflow conditions are considered upstream of the expansion: a laminar Poiseuille flow and a turbulent inflow. The underestimation of the reattachment location of the primary geometry-induced separation bubble leads to the spatial shift of the secondary pressure-induced separation bubble formed on the top wall in the laminar BFS. Despite this underestimation in iLES, an excellent agreement has been obtained on the mean flow properties and Reynolds stress budgets with 12 times fewer grid points than DNS.

Keywords-component—Implicit LES, Backward-facing step flow, Separated flows, Separation bubble.

I. INTRODUCTION

The occurrence of flow separation and reattachment plays an important role in the performance of a wide range of aeronautical and environmental engineering applications. These phenomena affect heat, mass, and momentum transport. As heat transfer and mixing rates increase, the performance of devices such as combustors and heat exchangers is improved. On the other hand, the generation of flow-induced vibrations due to velocity and pressure fluctuations, as well as flow resistance, reduces the efficiency of fluid-thermal devices.

Flow over a backward-facing step is of great interest regarding its common applications in engineering, simple geometry and complex flow dynamics, including separation, reattachment and recirculation regions. The sudden expansion at the step edge leads to a strong adverse pressure gradient, which causes the flow to separate from the wall, form a recirculation region and reattach to the wall downstream of the step. Numerous numerical and experimental studies have been made in the understanding of internal and external BFS flows during the past decades (e.g. [1]–[4]). Nonetheless, few studies have focused on the development of large-scale structures and low-frequency unsteadiness mechanisms in separated shear layers of transitional and turbulent regimes at high Re numbers (e.g. [5]–[7]). In experimental studies, the presence of low-amplitude disturbances in the inflow is inevitable and affects the laminar-to-turbulent transition process. Thus, numerical and experimental results differ in details, e.g. reattachment location, although similar general trends are obtained [8], [9].

DNS studies of BFS flows at high Re numbers require high computational resources. An early study by Armaly et al. [1] on a flow over a bounded BFS presented 2D experimental and numerical results over a wide range of Re numbers 70 < ReD < 8000, based on the hydraulic diameter of the inlet channel. Up to ReD = 400, 2D numerical results are matched with experimental data. Above ReD = 400, however, numerical predictions are discrepant from experimental results. They proposed that the flow lost its two-dimensionality nature due to perturbations induced by the formation of a secondary separation bubble on the top wall opposite the step. Thus, 2D numerical simulations can not accurately predict the 3D behaviour of the flow. They also found that in some ranges of Reynolds number, a tertiary bubble develops on the lower wall downstream of the primary separation bubble. Consequently, numerical studies should be conducted in a 3D domain that is sufficiently large to cover all flow structures and has a high temporal resolution to capture all time scales. To reach this goal, high computational resources are required. DNS studies, which require sufficiently fine grids, may not be the best method for solving BFS flows at high Reynolds numbers.
numbers. The LES technique is an alternative method that requires comparatively coarser grids and thus has moderate computational costs.

While classical explicit LES techniques (e.g. the Smagorinsky model) have been developed and are widely used in the simulation of flows, no general agreement has been reached on modeling subgrid scales. Moreover, when high-order numerical schemes are employed, subgrid-scale models may not act as a control, but as a source of numerical errors [10]. On the other hand, implicit LES relies on numerical treatments to supply dissipated energy from unresolved structures rather than employing any explicit model. This approach was first introduced as the Monotone Integrated Large Eddy Simulation (MILES) [11] and then evolved under the name iLES in several varieties, e.g. [12]–[14]. It has received increasing attention because of its extensive applications and accurate prediction of turbulent properties without the need for near-wall modeling. There are various approaches in iLES which are discussed comprehensively in [15]. Discretization of both convective and viscous terms in the Navier-Stokes equations are the sources of numerical dissipation. The majority of iLES approaches reinforce numerical dissipation by employing upwind schemes to discretize the convection terms [15]. Nonetheless, a controlled numerical dissipation through the discretization of the viscous term in the computation of second derivatives was proposed in [16]. This approach is further developed in a 3D simulation of Taylor Green vortex at a noticeably high Reynolds number [17]. Additionally, highly encouraging results have been achieved in other types of turbulent flows, such as impinging jets [10], wind turbine wakes [18] and bypass transition of a boundary layer [19].

The present paper focuses on the assessment of the iLES method for simulating flow separation and subsequent reattachment over a BFS. Both laminar and turbulent inflow conditions have been studied. Simulating a laminar BFS flow is challenging because it comprises laminar, transitional, and turbulent regimes. Moreover, both geometry-induced and pressure-induced separation bubbles are developed. Simulating a turbulent BFS is also crucial due to the need to generate an accurate turbulent inflow. Additionally, the flow is highly unstable. We conducted DNS for both cases to provide our own reference solution for comparison with the iLES results.

II. METHODOLOGY

A. Configuration

We consider an internal flow of an incompressible, Newtonian fluid with a constant density through a BFS channel. The schematic view of the flow geometry, shown in Fig. 1, indicates that the computational domain size is $40h \times 2h \times 6h$, where $h$ represents the step height which is equal to the inlet section height. The streamwise length upstream and downstream of the step is $L_s = 6h$ and $L_z = 34h$, respectively; a step length greater than $5h$ is sufficient to prevent the undesirable effects of the inlet on the flow passing through the step [3]. To ensure the development of flow structures along the spanwise direction, we set $L_z = 6h$, which is greater than the values used in prior studies [5], [20], [21]. The Reynolds number based on the step height $h$ and the inflow bulk velocity $U_b$ is $Re = U_bh/\nu = 5000$, which is as high as it can be while still being computationally feasible. The expansion ratio is defined as $L_y/h = 2$ where $L_y$ is the height of the channel downstream of the step. We generate a facing step inside the rectangular box domain using the immersed boundary method described in [22].

![Figure 1: Schematic of the flow configuration](image)

The origin of the coordination system is fixed at the mid-span of the bottom wall. The streamwise, wall-normal and spanwise directions are denoted by $x$, $y$ and $z$, respectively. The flow evolves in the streamwise direction. All parameters are non-dimensionalized with respect to the step height $h$ and the bulk velocity of the inlet flow $U_b$.

No slip boundary conditions are imposed on walls, including the step, top and bottom walls. Due to the assumption of statistically homogeneous flow along the spanwise direction, a periodic boundary condition is taken in this direction. At the outlet section, a convective boundary condition is applied to all velocity components, enabling vortices to move out of the domain without considerable disturbances [23]. The effects of the outflow convective condition are restricted within one step height before the outlet section [5]. Therefore, the region affected by the convective condition is negligible since the domain length is long enough.

Two case studies are explored, both of which have identical configurations and boundary conditions except for the inlet flows. In the first case study, hereafter denoted by "laminar BFS", the inlet boundary condition over the step is specified as a fully-developed laminar Poiseuille flow having a maximum streamwise velocity of $U_{max} = \frac{3}{2}U_b$.

In the second case study, hereafter denoted by "turbulent BFS", the inlet flow is turbulent. The generation of turbulent inflow begins by simulating the laminar BFS configuration, except that a small box is inserted close to the inlet section to trigger turbulence. The box is removed once reaching a fully-developed turbulent flow. The recycling method provided in [24] is then applied so that the velocity data from a specific $yz$-plane at $x = x_{rec}$ are recycled to the inlet section at each iteration. We choose the recycling length at $x_{rec} = 4.5h$. Since the mean flow is parallel, the recycling technique does not require any scaling, unlike in other types of flow such as an evolving boundary layer. Moreover, no interpolation is necessary since a structured and uniform Cartesian grid has been employed in all three directions.

ID 342 – CFD symposium
B. Numerical Methodology

The massively-parallel high-order finite difference flow solver “Incompact3d” is employed for both DNS as a reference and the iLES to numerically solve the incompressible Navier-Stokes equations with the aforementioned boundary conditions [25], [26]. Compact sixth-order finite difference schemes are used to compute the spatial discretization, including interpolations, first and second derivatives.

In the iLES technique, as previously noted, the viscous term is considered as a source of numerical dissipation, thereby the convective term should be discretized with the least possible error. The convection term is therefore expressed in the skew-symmetric form to minimize the aliasing errors that may emerge in the numerical solution [27] and conserve the kinetic energy. The latter implies that the kinetic energy dissipation only depends on the discretized viscous term. To discretize the viscous term, in the sixth-order difference scheme should be characterized as an over-dissipative scheme to provide an artificial dissipation. Extra-dissipation near the grid cut-off scales is selectively added by mimicking a Spectral Vanishing Viscosity (SVV) kernel, denoted by \( \nu_s \) [28]–[31]. The ratio of \( \nu_0/\nu_s \) specifies the level of extra-numerical dissipation where \( \nu_0 \) is the magnitude of the spectral viscosity at the cut-off wavenumber. According to a Pao-like solution suggested in [17], an optimal value for the ratio of \( \nu_0/\nu_s \) is estimated exclusively in the case of homogeneous and isotropic turbulence. Based on this method, \( \nu_0/\nu_s = 4.0 \) is chosen.

C. Spatial and temporal resolution

The mesh is generated in Cartesian coordinates with a half-staggered arrangement to prevent spurious pressure oscillations. The number of structured grid points in DNS is \( N_x \times N_y \times N_z = 1601 \times 601 \times 432 \), which are uniformly distributed in three directions. It corresponds to grid spacings in wall units of \( \Delta x^+ = 5.17 \), \( \Delta y^+ = 0.69 \) and \( \Delta z^+ = 2.87 \) at the outlet section. The mesh is finer compared to DNS studies of [5] and [21], so all spatial scales are resolved in the highly refined mesh. A low grid resolution is chosen for the iLES method. The uniform structured mesh domain includes \( N_x \times N_y \times N_z = 1025 \times 257 \times 128 \) grid points. The normalized grid spacings at the outlet section are \( \Delta x^+ \times \Delta y^+ \times \Delta z^+ = 7.54 \times 1.51 \times 9.05 \). Despite having a coarser mesh resolution than DNS, three nodes still exist in the viscous sublayer region \( (y^+ < 5) \) along the wall-normal direction to resolve near-wall motions. In both calculations, the time step is set at \( \Delta t = 5 \times 10^{-4} t_{\text{b}} / U_{\text{b}} \), which is small enough to capture all temporal scales of the flow.

III. RESULTS

A. Laminar BFS

Figure 2 shows the overall structure of the flow using the time- and spanwise-averaged flow obtained by DNS and iLES computations. The generation of separation bubbles is qualitatively similar in both simulations. The contours of streamwise velocity show a significant portion of reverse flow within the separation bubbles. Furthermore, the streamlines confirm the existence of recirculation regions. A geometry-induced separation bubble (SB1) is generated due to the sharp edge of the facing step on the lower wall. There is also a smaller bubble inside the SB1, at the corner of the facing step and adjacent to the wall, which has also been observed in other works, e.g., [3], [21]. Moreover, a secondary separation bubble (SB2) is induced by the presence of a strong adverse pressure gradient on the upper wall. A very small inner bubble is also found inside SB2. Two small bubbles certify that the resolution of grids is sufficient to capture small eddies in both DNS and iLES computations. However, in this study, we mainly focus on the two large separation bubbles (SB1 and SB2) and the main flow passing between them. The red dot-dashed lines correspond to the isocontour \( (U = 0) \), which passes through the centers of recirculation regions, separates the forward and reverse flows, and helps to visually identify the mean separation and reattachment points.

|          | \( x_{r1} \) & 26.07 & 20.39 & 30.38 & 14.07 & 10.01 |
|----------|-----------------|---------|----------|---------|---------|
| DNS      | 7.7             | 22.7    | 17.1     | 27.17   | 10.7    | 10.07    |
| iLES     | 7.7             | 22.7    | 17.1     | 27.17   | 10.7    | 10.07    |

TABLE I: Location of the separation and reattachment points of the mean flow in DNS and iLES

The scatters, in Fig. 2 denote mean separation and reattachment points that are summarized in Table I for both DNS and iLES. There is a significant difference regarding the location of the reattachment point \( x_{r1} \) on the bottom wall so that in the iLES \( x_{r1} \) is about 14.8% smaller than in the DNS. This discrepancy at the bottom wall leads to the spatial shift of the separation and reattachment points at the top wall, while the difference in the mean length of SB2, obtained by DNS and iLES, is less than 0.6%. The undesirable oscillations caused by numerical errors in the coarse grid reinforce the instability of the separated shear layers downstream of the facing step. Higher instability leads to an earlier transition from laminar to turbulent flow, and hence an earlier reattachment. The reattachment location of SB1 is therefore underpredicted in the iLES simulation with a coarse grid. Additionally, the separation and reattachment locations of SB2 shift upstream in space. Schäfer et al. (2009) [21] conducted two DNS simulations on an internal BFS flow using fine and coarse mesh resolutions, with \( \text{Re} = 3000 \) and \( \text{Er} = 1.9423 \). The comparison of the mean results obtained from the DNS conducted using a fine grid to our DNS results revealed a good agreement, with less than 1.4% difference observed in the reattachment location of SB1. However, this difference can be attributed to the difference in Re number and expansion ratio used in the simulations. The authors examined the sensitivity of BFS flows to grid-induced oscillations by comparing the results obtained from fine and coarse grid simulations. They found that the reattachment location of the primary separation bubble was underestimated in the coarse grid simulation, with a difference of around 3.64%. On the other hand, the separation bubble length on the top wall was overpredicted on the coarse mesh simulation, resulting in a
Figure 2: Mean streamlines (solid black lines) of the laminar BFS flow with contours of time- and spanwise averaged streamwise velocity (a) DNS (b) iLES. Dot-dashed red lines denote zero mean velocity.

Figure 3: Skin friction coefficient along the (a) bottom wall and (b) top wall.

length that was 19\% greater than that obtained by the fine mesh simulation. Therefore, the errors found between iLES and DNS results in this study are less than those identified in [21]. It can be noted that the ratio of grid points between their two DNS simulations is less than 2, while the number of mesh points in iLES is 12 times less than in DNS. The iLES approach utilizes artificial viscosity to effectively damp grid-induced oscillations in turbulent and transitional regimes. However, it is not entirely effective in damping spurious disturbances in laminar regions. Perrin & Lamballais [19], who simulated a bypass transition of a boundary layer, declared that the iLES model became active in transitional and turbulent regions while having a modest impact in laminar regions.

Figure 3 depicts the evolution of the top and bottom wall skin friction coefficients in the streamwise direction from the time- and spanwise-averaged flow obtained by DNS and iLES simulations. The skin friction coefficient is defined as $C_f = \tau_w/(\frac{1}{2} \rho U_b^2)$, where $\tau_w$ is the time- and spanwise-averaged wall shear stress. To compare the results on each wall, the corresponding separation point positions are subtracted from the streamwise position and divided by the corresponding separation length $(x - x_s/L_b)$. The top and bottom $C_f$ obtained by iLES agree very well with the DNS results. They exhibit the same behaviour and have roughly similar minimum and maximum values. The location of separation and reattachment points, where $C_f$ reaches zero, are matched with those observed in Fig. 2. On the bottom wall (Fig. 3a), $C_f$ is almost zero in the first 50\% of SB1 $(x < 16)$, known as a 'dead-air' region, at which the velocity is very low. The magnitude of $C_f$ becomes negative in the recirculation region to reach a global minimum of $C_f \simeq -0.0022$, which occurs upstream of the reattachment point. A rapid rise is observed immediately after the minimum peak due to the reattachment of the shear layer to the wall, followed by a slight decrease because of the presence of SB2. The skin friction coefficient on the top wall (Fig. 3b) attains a local minimum past mean separation point, followed by a weak increase in amount. The skin friction reaches zero and changes sign in
the range of \(0.91 < (x - x_{c2})/L_{c2} < 0.98\). This supports the presence of an inner bubble inside SB2, which contains an anti-clockwise flow. iLES results successfully detect this small bubble contained within SB2, which is a complex behaviour in the flow.

### B. Turbulent BFS

Mesh grids and all simulation parameters in turbulent BFS are the same as in laminar BFS, as explained in section II. Figure 4 shows the contours of the time- and spanwise-averaged streamwise velocity obtained by both the iLES and DNS methods. The upstream turbulent flow separates at the edge of the step due to the sudden expansion of the geometry. The separated shear layers reattach to the wall far downstream, forming a separation bubble. The mean flow distribution exhibit close agreement with the DNS data of turbulent BFS flow from [4], despite the fact that Re number in their study is slightly higher (Re = 5600). A qualitative comparison between iLES and DNS reveals that the topology of the main flow, the geometry-induced separation bubble and the small inner bubble at the corner of the step are highly consistent. Moreover, the mean flow distribution exhibits close agreement with the DNS data of turbulent BFS flow from [4], despite a slightly higher Re number in their study (Re = 5600). The mean reattachment location of the separation bubble, denoted by circle marks in Fig. 4, is at \(x_r = 13.64\) in the iLES computation and \(x_r = 13.73\) in the DNS. The error percentage is quite small, about 1%. Also, the length of the inner bubble in both simulations is 1.8 \((x_p = 7.8)\).

Besides the mean flow, the Reynolds stresses at three distinct streamwise locations, upstream of \(x_r\) at \(x = 10\), close to \(x_r\) at \(x = 15\) and far downstream of \(x_r\) at \(x = 20\) are compared in Fig. 5. At \(x = 10\), all Reynolds stresses are in perfect agreement. At \(x = 15\) and \(x = 20\), both \(\overline{u'v'}\) and \(\overline{u'v'}\) are consistent along the wall-normal direction, whereas \(\overline{u'v'}\) has a small mismatch in the middle of the channel height, especially at \(x = 15\). This difference can be attributed to the grid-induced oscillations in highly unstable shear layers. Thus, despite the slight differences between iLES and DNS findings, iLES computation is successful, given that the total number of nodes in iLES is 12 times less than in DNS.

### IV. Conclusion and future works

To have a moderate computational cost in the simulation of a separated flow over an internal BFS, the iLES approach in a relatively coarse grid was evaluated. The iLES method relies on the introduction of an artificial numerical dissipation through the discretization scheme of the viscous term. Two inlet conditions, laminar and turbulent, were analyzed. To provide reference data, the same configurations were simulated using DNS. In both case studies, the mean flow behaviour obtained by DNS and iLES was in excellent qualitative agreement. In the laminar BFS, a geometry-induced separation bubble developed behind the step, followed by a pressure-induced separation bubble on the upper wall. The iLES approach underestimated the reattachment location of the primary separation bubble while predicting the length of the secondary separation bubble with less than 1% error. In the turbulent BFS, the recycling method demonstrated the ability to successfully generate turbulent inflow. The iLES approach accurately predicted the geometry-induced separation bubble length and Reynolds stress budgets. Previous studies have shown that DNS simulation of BFS flows is extremely sensitive to grid-induced numerical errors. However, in this study, we explored that the iLES technique can effectively improve the predictions in coarse grid simulations by employing an extra-numerical dissipation to damp the grid-induced oscillations. The underestimation of the reattachment length of the primary separation bubble in laminar BFS can be attributed to the lower effectiveness of this model in laminar regions compared to transitional and turbulent regimes. This approach has the potential to be improved by recent developments in the ILES approach like spatially-varying artificial viscosity [32] and hyperviscous filtering [33].

### REFERENCES


Figure 4: Mean streamlines (solid black lines) of the turbulent BFS flow with contours of time- and spanwise averaged streamwise velocity (a) DNS (b) iLES. Dot-dashed red lines denote zero mean velocity.

Figure 5: Comparison of Reynolds stresses at (a) $x = 10$, (b) $x = 15$, (c) $x = 20$. Solid black lines represent iLES results and symbols denote DNS results.