Progress towards simulation of Krueger device motion with Lattice Boltzmann Methods

J Ponsin, C Lozano

Computational Aerodynamics Group, National Institute of Aerospace Technology (INTA). Carretera de Ajalvir, km4, Torrejón de Ardoz 28850, Spain
ponsinj@inta.es

Abstract. This paper presents preliminary results of the application of a Lattice-Boltzmann code to the prediction of the unsteady aerodynamics present during the deployment/retraction of a Krueger device. The present study has been performed within the EU-Project UHURA, which is dedicated to the validation of URANS and other unsteady flow simulation methodologies for the prediction of the flow on high lift devices designed for laminar wings. Several case setup issues are examined and verification with RANS is presented. Lessons learned during the first phase of the project are drawn.

1. Introduction
Laminar flow wing technology requires specific high lift concepts that are subject to the main constraint that in cruise conditions, and with the device in retracted position, the flow on the upper surface must remain unperturbed. One possible solution to replace the classical slat element is to use a folded Krueger device [1]. The numerical simulation of the deployment/retraction of a Krueger device poses numerous challenges regarding physical and numerical modelization. The flow has to be computed in presence of large enforced motions, which requires special numerical techniques. There are large turbulent separated flow areas downstream of the Krueger device during a significant fraction of the device motion. Such a large, unsteady separation could affect significantly the flow over the rest of the elements. Therefore, the accuracy of the CFD tools must be investigated.

The EU funded UHURA project aims at performing experimental research on the unsteady complex flow behaviour associated to the deployment/retraction of a Krueger device, as well as improving and validating unsteady CFD simulation methodologies. The focus is set on URANS solvers and the special grid techniques required to dealing with large element motions (chimera, immersed boundaries, etc). Additionally, other unsteady flow methodologies are investigated, such as advanced hybrid RAN-LES turbulence models as well as alternative CFD techniques such as the Lattice Boltzmann Method (LBM).

This paper presents a preliminary study of the application of the LBM methodology to the prediction of Krueger device induced unsteady aerodynamics. This study has been performed in the first phase of the UHURA project with the aim of finding guidelines for best practices and possible limitations of the LBM. The knowledge gathered will be subsequently used in the validation phase of the project.

2. LBM Computational Methodology
The LBM is a promising alternative method for the CFD simulation of unsteady flows due to several reasons: first, it is intrinsically formulated for unsteady flows and has very low numerical dissipation/dispersion errors, which makes it suitable for joint use with high-fidelity turbulence models
based on scale resolving methodologies. Second, its underlying adaptive Cartesian octree mesh approach, together with the way wall boundary conditions are imposed, is particularly well suited to dealing with complex and/or moving geometries. Finally, the approach is computationally extremely efficient (even though the time step advancement is limited by the explicit stepping nature of the time integration method). The LBM is a mesoscopic CFD approach that is based on numerically solving a discretized version of the Boltzmann transport equation for the probability distribution function of particles at a given spatial location and velocity. Macroscopic flow properties such as density, velocity and pressure, arise as the successive (velocity) moments of the distribution functions. It can be shown theoretically that the isothermal weakly compressible NS equations are obtained from a Chapman-Enskog ( multiscale) expansion of the moments of the LBM equations.

2.1. Numerical details of the LBM solver
Within the UHURA Project, INTA is using the LBM code DS-Xflow [2] for assessing LBM as an alternative CFD method to URANS. Xflow is based on a three dimensional 27-velocity lattice model (D3Q27), which seems to be the most appropriate set of velocities for turbulent computations due to the higher level of symmetry of the velocity lattice. The collision operator is based on a central moment model that has a high degree of accuracy and stability, and benefits from an extended Mach number application range compared to standard collision operators. The meshing approach is based on an adaptive Octree Cartesian technique, where the number and arrangement of Octree levels depends on the far-field and minimum resolutions specified in the geometry as well as the prescribed buffer layers among different levels. On-the-fly adaptive mesh refinement is used in dynamic motions. The mesh is adapted in the near wall region to guarantee sufficient resolution for the wall turbulence model. In addition, a refinement/de-refinement adaptive algorithm based on the local level of vorticity has been used to resolve the wakes and large separation areas that appear during the motion of the Krueger device. Presently, these two adaptive options are only available for parallel shared-memory architectures, which supposes a limitation for the mesh sizes used in the simulations that must be borne in mind.

The DS-Xflow software uses two alternative algorithms that allow unsteady flow simulations for arbitrary geometry motions. This is the main reason for choosing this software for the assessment of LBM methods within UHURA. More details about the used methodology will be provided in section 4.

2.2. Turbulence Model
DS-Xflow uses a LES approach for turbulence simulation. The SGS model used in our simulations is the wall-adapting local eddy viscosity (WALE) model. Since the mesh and time resolution required for wall resolved LES are not affordable nowadays, at least for the Reynolds numbers considered in the project, a wall-modelled LES (WMLES) approach is used in which only the outer portion of the boundary layer is fully resolved by the LES, while the effect of the near-wall region is modelled. Xflow uses a wall-shear stress approach to model the effect of the near-wall turbulence, i.e. the wall boundary condition is prescribed by using the wall shear stress information obtained through an algebraic wall function (e.g. the classical law of the wall) by matching the LES velocity information at a certain location called the exchange location. A non-equilibrium wall function is used that accounts for the effect of strong pressure gradients and separated flows.

3. Computational setup
The objective in the first phase of the UHURA project, regarding the LBM alternative method, is to establish a set of best practices guidelines for the simulation of the Krueger device that could be used later in the validation phase of the project. In a first step, meshing strategies and numerical settings have been studied on static cases where the geometry is fixed at representative positions of the deployment/retraction movement.

An initial design of the Krueger device configuration was provided by DLR for this preliminary study. It consists of the laminar leading edge DLR-F15-LLe high lift section supplemented with an initial Krueger device design. The flow conditions selected for the study are Re = 2 Mill and Mach = 0.15 that correspond, approximately, to the operating conditions in the ONERA L1 and DNW-NWB wind tunnels. The angle of attack, $\alpha = 10^\circ$, is chosen to be representative for the approach and landing
flight parameters. In order to show the flow features that appear during flap deployment/retraction, four positions of the Krueger device have been defined as test cases: (1) Krueger device in fully retracted position, (2) almost perpendicular to the incoming flow, (3) leading edge passing and (4) fully deflected position. These positions have been obtained by applying simple kinematic laws (constant rotation velocity around the provided hinge points).

3.1. Grid details

The WMLES approach requires 3D meshes, and is particularly sensitive to the near-wall mesh resolution. Hence, special care is needed for mesh generation. Even though the LBM approach is computationally quite efficient for scale resolving simulations, the final target in the project is to perform the computations for dynamic cases (complete deployment/retraction of the Krueger device) and this implies also a limitation on the mesh sizes in order to keep the required computational time as low as possible.

The computational domain consists of a box of dimensions $L_x = L_y = 100 \times C_{ref}$, where $C_{ref}$ is the chord of the main wing in fully retracted position. The span-wise size, $L_z$, has been prescribed to approximately twice the Krueger Flap length, taking into account that (1) it should be large enough to contain all the larger 3D turbulent structures produced in the separation region behind the Krueger device and (2) it should be limited by the resulting size of the mesh. (Bear in mind that the Cartesian meshes used in the LBM are isotropic in nature).

Since the resolution of the outer boundary layer ($y / \delta \geq 0.2$) is the target of the WMLES approach, the relevant scale for mesh generation at the wall region is the turbulent boundary layer thickness, $\delta$. The key parameter in WMLES is the number of points per boundary layer thickness $n / \delta$. Several studies indicate that the recommended values for this parameter lie within the range $10 \leq n / \delta \leq 30$. This is the main guideline when prescribing the mesh resolution at wall regions. Another restriction for the Xflow WMLES is the normal distance to the wall of the exchange location. For the shear stress-based approach, it is recommended that this point be located within the log layer (whenever it exists). In addition, different resolutions at each geometrical component of the high-lift system have been prescribed to capture the strong pressure gradients at the leading edges. The wake resolution has been prescribed by using the shear layer thickness scales obtained from reference RANS resolutions. Details of the resolutions employed can be found in [3]. Figure 1 shows an example of the resulting mesh for the perpendicular position of Krueger device.

3.2. Computational details

The boundary conditions prescribed for the LBM computations are velocity inlet at the upstream plane, pressure outlet at the downstream plane and periodicity for the lateral walls. We have adopted the acoustic scaling approach to perform the LBM computations, which fixes the time step in terms of the prescribed Mach number, reference velocity and spatial resolution as $\Delta t = M \Delta x / \sqrt{3} U_{ref}$, with the additional requirement that the ratio $\Delta t / \Delta x$ must remain constant across refinement levels. The dimensionless time step for the finest spatial resolution (a 10th Octree level) used in this study is about $\Delta t U_{ref} / C_{ref} \approx 5.4 \times 10^{-5}$. Static simulations were run for over 15-20 convective time units (depending on the test case), after which statistics were collected for over 5-10 convective time units. Table 3 shows details of the grids and computations.

One of the strong features of the LBM is its computational efficiency for scale resolving simulations. Table 5 shows the maximum sizes of the meshes reached during static computations, the wall-normal and spanwise resolutions (referenced to the cell size at the wall, $\Delta$) and the computational cost of 15 flow passes in KCPUs. The boundary layer thickness, which is used to measure the wall-normal resolution, is estimated by the thickness at the trailing edge of the main wing of a 2D RANS boundary layer.
Figure 1. View of the mesh with dynamic mesh adaptation at the wake area for the perpendicular position of the Krueger device.

Table 1. Mesh and computational cost for details for 15 flow passes.

|                | Millions of cells | $\delta/\Delta$ | $L_z/\Delta$ | Wall-clock KCPUh |
|----------------|-------------------|-----------------|--------------|------------------|
| Retracted      | 14.3              | 16              | 256          | 1.584            |
| Deployed       | 11.3              | 10              | 256          | 2.592            |
| Perpendicular  | 20.2              | 10              | 256          | 2.208            |
| Passing LE     | 19.2              | 10              | 256          | 2.113            |

4. Results
Experimental results are not available at this stage of project, so in order to have reference results for verification of the LBM computations and to guide the mesh generation for WMLES, several 2D RANS computations have been carried out. Cases 1 and 4 (retracted and fully deployed KF) have been selected for computation since the flow for these cases is expected to be attached and thus RANS results are supposed to be fairly accurate. Two RANS solvers, the DLR-TAU code [4] and FLUENT, and two turbulence models, SA and K-omega SST, have been used on different meshes, ranging from near-wall to wall function resolutions.

4.1. Comparison with RANS
$C_p$ distributions obtained with LBM and RANS are compared in figures 2 and 3 for the fully retracted and deployed configurations, respectively. There are significant discrepancies, especially on the suction sides, due to the lack of developed turbulence in the boundary and shear layers for the LBM simulations. Large differences in $C_p$’s are found, especially for the retracted case, where the high adverse pressure gradient significantly thickens the boundary layer downstream of the suction peak and this should have an impact on the $C_p$’s at the rear part of the main wing. Furthermore, the suction on the upper side of the rear flap is largely over-predicted by the LBM code due to the lack of turbulent mixing in the shear layers, which significantly affects the velocity and pressure field at the flap region.

Lift coefficient values for the retracted/deployed configurations are shown in figure 10. (Note that in LBM, the forces are usually obtained by the momentum exchange method). The results for the deployed configuration are misleadingly good, since they are produced as a result of cancellation errors. For the retracted configuration, on the other hand, the lift errors are quite large, partly due to the over-prediction of the lift of the flap.
Instantaneous flow visualization is presented in figure 4, where the iso-surfaces of the dimensionless Q-criterion are used to show the resolved turbulent structures for the four computed positions. It can be readily seen from the plot that in the retracted and deployed positions significant (resolved) turbulent content is only present downstream the trailing edge of the rear flap, but not at the rest of the configuration. Therefore, the turbulent boundary layers and wakes are not properly simulated.

For the perpendicular Krueger device position, there is a geometry induced massive separation behind the device which produces resolved turbulent content downstream, but on the upper surface turbulence is still lacking.

In the passing leading edge position, turbulent content is developed on the upper wing section due to turbulence fluctuations coming from a separation produced at the bull nose-panel junction. It can be concluded that for the present 2.5D simulations, one has to introduce some disturbances into the flow in order to activate turbulence in the outer boundary layer, at least for static computations, since otherwise the available WMLES model does not properly resolve the turbulence.
4.2. Turbulence tripping study

It is clear from the above results that, at least for static 2.5D simulations, it may be necessary to perturb the flow in order to obtain resolved turbulence in the boundary layers. An exploratory study has been carried out to find out whether introducing perturbations via geometrical roughness elements can effectively activate the WMLES in the outer part of the boundary layer and to assess their overall impact on the flow. The roughness elements we have tested are inspired on the work of [5], where cylindrical roughness elements are used to promote a rapid switch between RANS and WMLES zones (i.e to introduce self-sustainable resolved turbulence in the outer boundary layer). An array of small cylindrical roughness elements is used as vortex generators that produce stream-wise counter-rotating vortices. Thanks to the lift-up effect, these streamwise vortices become unstable as they evolve downstream, producing sustainable turbulence.

The roughness elements are characterized by the height $h$ and diameter $d$ of the cylinders, as well as the span-wise spacing between consecutive cylinders $\lambda_z$ (see figures 5 and 6). All the parameters are referenced to the outer scale $\delta$ of the unperturbed (RANS) boundary layer. A parametric study has been carried out within the project. Figure 7 shows the $Q$ parameter for an array of roughness elements located at 20% of the chord and with height of the order of $\delta$ and spanwise spacing $2\delta$. It can be seen that turbulence is quickly promoted, and hairpin structures typical of the outer part of the boundary layer are clearly visible. Figures 8 and 9 show some sample results obtained in this study. The prediction of the pressure coefficient distribution has clearly improved at the rear part of the wing and on the upper side of the flap. The predicted averaged velocity profile at the rear part of the main wing is also improved, although differences with RANS results are still significant. As a consequence of the improved $C_p$ prediction, the lift values are also improved as shown in figure 11.

5. Dynamic computations

One of the strong potential advantages of the LBM, and which makes it attractive for the type of problem addressed in UHURA, is the ability to deal with complex geometries and moving objects in a computationally efficient and robust way. The common approach within the LBM framework is to use
immersed boundary conditions or similar approaches. In this regard, as far as we know, the code used in the present study, DS-Xflow, is the only commercial solver with capabilities for arbitrary body motion.

**Figure 8.** Cp distribution at a spanwise section of the main element amidst two roughness elements.

**Figure 9.** Averaged streamwise velocity profiles at $x/c = 0.88$ on the upper surface of the main element.

**Figure 10.** Lift coefficient for the retracted/deployed device without tripping elements.

**Figure 11.** Influence of turbulence tripping for the retracted Krueger device.

Simulation. The code has two algorithms to deal with moving objects, but the so-called immersed moving boundary method was selected after several tests due to its better computational efficiency. Basically, this algorithm is based on a modified partially saturated cell method (PSM) which amounts, roughly speaking, to a modification of the collision operators with an additional term proportional to the solid volume fraction of the cell. In our own experience, the PSM algorithm has a 40% CPU time overhead with respect to static computations (fixed geometry).

Based on the computational setting described in section 3 for static cases, a set of dynamic computation tests have been carried out. The aim was to address several questions regarding computational performance in dynamic simulations, checking the prescription of external kinematic laws, the behaviour of turbulence during the motion, as well as to have a first glimpse of the time evolution of forces. A typical sequence for Krueger device motion was the following: the initial configuration was simulated for about 25-30 $C_{ref} / U_\infty$ time units in static position to achieve proper flow stabilization, followed by the deployment/retraction phase and a second stabilization phase of about 7-10 $C_{ref} / U_\infty$ time units in the final static position. Complete full deployment/retraction cases (with a total duration of 1 s) were run on a mesh with 11 million cells on 24 cores with a time step on the finest grid level of $1.8 \times 10^{-6}$ s. The simulations required around 11 days (only the motion phase) of continuous runtime (roughly 6.4 wall clock KCPUh). As an example of the preliminary results with dynamic
geometries, figure 12 shows a sequence of pictures of the velocity field during the Krueger device retraction. It can be seen that, in contrast to the static case, a turbulent wake is present on upper side of the flap and this is reflected on the final lift values and pressure distribution obtained at the end of the motion (not shown). Therefore, dynamic computations could have a beneficial effect, introducing turbulent fluctuations during the motion that activate the WMLES model.

![Velocity field magnitude visualization at different stages of the Krueger device retraction.](image)

**Figure 12.** Velocity field magnitude visualization at different stages of the Krueger device retraction.

6. Summary
In a first phase of the UHURA project, a WMLES based LBM numerical code has been assessed for the prediction of the unsteady flow present during the motion of a Krueger device designed for laminar wings. Best practices guidelines in terms of grid resolution and computational settings have been studied. First results indicate that this method shows a great potential in terms of computational efficiency, mesh generation and robustness for dealing with moving geometries, although more studies are needed to validate the physical models used for turbulence. A LBM validation activity will be performed in the second phase of the project using the results obtained in wind tunnel testing within the project.

References
[1] Iannelli P, Wild J, Minervino M, Moens F and Raets M 2012 Analysis and applications of suitable CFD–based optimization strategies for high lift system design. *ECCOMAS 2012 proceeding on CD-ROM, paper 2833.*
[2] Trapani G, Brionnaud R, and Holman D 2018 Xflow contribution to the third High-Lift prediction workshop *AIAA paper 2018-2847, 2018 AIAA Applied Aerodynamics Conference.*
[3] Ponsin J, Lozano C 2020 Technical report on assessment of LBM and best practices for the prediction on unsteady aerodynamics during deployment/retraction phases of high lift systems. *UHURA Deliverable D22-2.*
[4] Gerhold T 2005 Overview of the Hybrid RANS code TAU. In: Kroll N., Fassbender J.K. (eds) *MEGAFLOW - Numerical Flow Simulation for Aircraft Design*, pp 81-92. Notes on Numerical Fluid Mechanics and Multidisciplinary Design (NNFM), vol 89. Springer, Berlin, Heidelberg. [https://doi.org/10.1007/3-540-32382-1_5](https://doi.org/10.1007/3-540-32382-1_5).
[5] Deck S, Weiss P, Renard N 2018 A rapid and low noise switch from RANS to WMLES on curvilinear grids with compressible flow solvers, *J. Comput. Physics* 363, 231-255

Acknowledgments
The project leading to this conference paper has received funding from the European Union’s Horizon 2020 Research and Innovation programme under grant agreement Nº 769088.