

Local Error Analysis and Comparison of the Swept- and Intersection-Based Remapping Methods

Matej Klima¹, Milan Kucharik^{1,*} and Mikhail Shashkov²

¹ Faculty of Nuclear Sciences and Physical Engineering, Czech Technical University in Prague, Břehova 7, Praha 1, 115 19, Czech Republic.

² XCP-4 Group, MS-F644, Los Alamos National Laboratory, Los Alamos, NM 87545, USA.

Received 12 December 2015; Accepted (in revised version) 17 July 2016

Abstract. In this paper, the numerical error of two widely used methods for remapping of discrete quantities from one computational mesh to another is investigated. We compare the intuitive, but resource intensive method utilizing intersections of computational cells with the faster and simpler swept-region-based method. Both algorithms are formally second order accurate, however, they are known to produce slightly different quantity profiles in practical applications. The second-order estimate of the error formula is constructed algebraically for both algorithms so that their local accuracy can be evaluated. This general estimate is then used to assess the dependence of the performance of both methods on parameters such as the second derivatives of the remapped distribution, mesh geometry or mesh movement. Due to the complexity of such analysis, it is performed on a set of simplified elementary mesh patterns such as cell corner expansion, rotation or shear. On selected numerical tests it is demonstrated that the swept-based method can distort a symmetric quantity distribution more substantially than the intersection-based approach when the computational mesh moves in an unsuitable direction.

PACS: 47.11.-j, 02.60.Ed

Key words: Conservative interpolation, remapping, numerical error analysis, swept regions, polygon intersections.

1 Introduction

For numerical simulations of fluid dynamics, the computational methods are typically categorized into two classes – the Eulerian and Lagrangian methods. In the pioneering work [11], the authors developed a more general framework combining best properties

*Corresponding author. *Email addresses:* klimamat@fjfi.cvut.cz (M. Klima), kucharik@newton.fjfi.cvut.cz (M. Kucharik), shashkov@lanl.gov (M. Shashkov)

of both Lagrangian and Eulerian approaches. This framework has been termed Arbitrary Lagrangian-Eulerian or ALE and since that, many authors have contributed to the investigation of its robustness, accuracy, or efficiency, see for example [1,3,10,15,24,25,29].

The ALE algorithm is usually separated in three distinct stages: (1) a Lagrangian stage, in which the fluid quantities and the computational mesh are advanced in time; (2) a rezoning stage, in which nodes of the (potentially distorted) computational mesh are moved to more optimal positions with respect to their geometrical quality; and (3) a remapping stage, in which all fluid quantities are conservatively transferred from the Lagrangian mesh to the rezoned one. The ALE algorithm preserves the advantages of the Lagrangian methods (such as low dissipation at the discontinuities or the computational mesh intrinsically following the fluid), while its Eulerian part (rezoning and remapping) prevents the computational mesh from degeneration often appearing in purely Lagrangian simulations. In this paper, we focus on the last part of the ALE algorithm – the remapping stage in single-material simulations.

In case of close computational meshes with the same topology, the remapping process can be formulated in a flux form, using fluxes of the involved quantities between cells which share the face. The fluxes are constructed by integrating the particular fluid quantity over certain transport volumes. Here, we discuss two methods of constructing such volumes [19,21,26,27]. The first method is more intuitive and employs the exact intersections of the computational cells with their neighbors in the new mesh. It is known to be more demanding in terms of computational resources, but it can, under certain circumstances, perform better in terms of solution symmetry (especially in case of discontinuous solutions and corner coupling), such as observed for example in [5,14]. The second widely used method approximates the calculated inter-cell fluxes using regions swept by the cell edges during the transformation from one mesh to another. It is robust and computationally less demanding, however, the approximation used raises concerns about its accuracy. The question is, whether it is possible to determine which method is better suited for a specific application or problem.

As far as we know, there exist two papers addressing the theoretical error analysis of the remapping methods. An error analysis based on the Fourier decomposition of the numerical error was performed in [23], showing that the faster swept-region-based method can under certain circumstances provide better results than using the intersections. Another analysis was presented earlier in [27], confirming the second order of accuracy and other properties of both methods. In the current paper, we theoretically analyze the numerical error of both methods locally, while treating the remapping process as an interpolation method rather than as a fluid flow through the computational cell edges.

However, to be able to determine which method is more suitable for specific data, an analysis of the overall accuracy for the entire computational domain is not sufficient. We need to perform an analysis of the distribution of the remapping error caused by each of the two methods. Either the fluid quantities, their derivatives (including the second derivatives), or the geometrical characteristics (such as cell volume or nodal movement