

15th Annual CFD Symposium, August 9-10, 2013, Bangalore

THE PROCESS OF GENERATING SINGLE LARGE COMBINED CLOUD FOR GRID-FREE SOLVERS

C.K.Niranjanan*
ckniranjanan@nal.res.in

KeshavMalagi*
Keshav.malagi@gmail.com

Kaushik Kumar Nagarajan*
kaushikkn@gmail.com

V.Ramesh*
vr@ctfd.cmmacs.ernet.in

S.M.Deshpande#
smd@jncasr.ac.in

* National Aerospace Laboratories, Bangalore, Karnataka, India.

Jawaharlal Nehru Center for Advanced Scientific Research, Bangalore, Karnataka, India.

ABSTRACT

Dealing with moving body problems, where one component moves relatively with respect to other, is a difficult task in CFD due to the efforts needed in grid handling for every delta change in position of the moving component. The inherent nature of mesh-free solvers reduces the efforts needed for these kinds of problems by operating on a cloud of points rather than a grid. A new method to handle moving body problems is proposed, where individual clouds are generated around each component and are combined into a single large combined cloud. The proposed method is applied to store separation problem and results generated using NAL-MCIR mesh-less solver is compared with experimental results.

Key words: Mesh-less solver, store separation, combined cloud, overlapping cloud, connectivity.

INTRODUCTION

A plenty of moving body problems like store separation from an aircraft, simulation of effects due to control surface movements, booster separation from a space launch vehicle, etc. exist, requiring the CFD community to handle these kinds of problem efficiently and effectively. Various CFD techniques have been demonstrated for handling moving body problems, including overset grid techniques[1] and Cartesian embedded-boundary method[2]. The overset grid uses a set of independent, overlapping grids and allows for grid motion without stretching or re-gridding. But an artificial boundary is formed between the overlapping grids and throughout the solution process there exist multiple grids communicating with each other through interpolation at the artificial boundary. The Cartesian embedded-boundary method uses Cartesian cut-cell method for automatized grid generation over the complex geometry, but re-gridding is required for every delta change in position or orientation of the moving component. The current work tries to establish the effectiveness of mesh-less methods in handling moving body problems by demonstrating a technique called “combined cloud generation technique” which is used to generate a single large combined cloud from multiple overlapping components’ clouds. The combined cloud generated can be used as the input to NAL-MCIR mesh-less solver. This technique allows for the component clouds to be moved without requiring regeneration of clouds and also the end product is a single cloud encompassing the entire domain.

FLOW SOLVER

The NAL-MCIR mesh-less solver, used in the current work, is an Euler solver based on least square kinetic upwind method. It uses a dissipation control function to reduce dissipation and to increase the order of accuracy[3, 4, 5]. The point cloud data input to the solver consist of an array of points along with the connectivity data for each of these points, where connectivity data of a point is the list of its neighboring points. Point cloud is generated by taking only the nodes of the grid generated by a grid generation tool.

COMBINED CLOUD GENERATION TECHNIQUE

In this technique a main stationary component is identified and cloud is generated around it encompassing the entire flow domain. Individual overlapping clouds are generated around all the other moving components, initially at their starting position. The moving components' clouds need not cover the entire flow domain. Whenever a component moves, the respective cloud moves along with it. An imaginary, closed bounding surface is formed around each of the moving component enclosing the entire component in it. The bounding surface can be of any arbitrary shape, subject to existence of some means to separate out points lying inside it from those lying outside it. Points belonging to stationary component cloud and lying inside the bounding surfaces of various moving components' bounding surfaces are deleted from the cloud and points lying outside of the bounding surfaces are retained. On contrary, points originating from moving components' cloud, and lying outside of their respective components' bounding surface are deleted from the components' cloud and points lying inside are retained. The deletion of points from stationary cloud creates holes in the flow domain. These holes are filled with the respective trimmed, moved moving components' cloud. The connectivity data of all the points lying near imaginary bounding surfaces is altered to account for the deletion process and to ensure that all the moving components' clouds are properly stitched to the stationary component cloud (i.e.) connectivity of a point in a moving components' cloud, near bounding surface, should contain points from stationary component cloud and vice versa (shown in fig-3). The process of combined cloud generation applied to a store separation problem is shown in fig-1 and the resulting combined cloud is shown in fig-2.

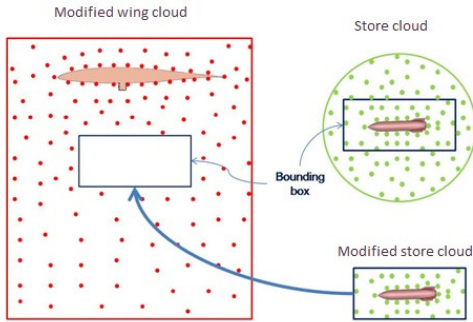


Fig. 1. Combined cloud generation technique

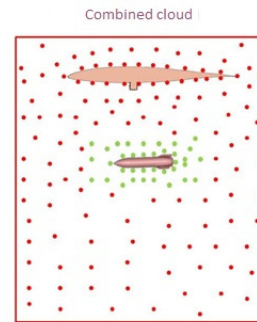


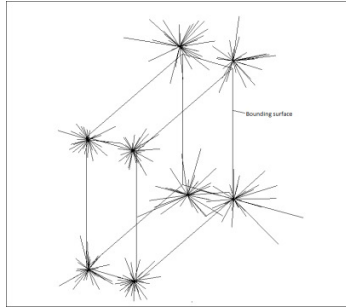
Fig. 2. Final combined cloud

TEST CASE

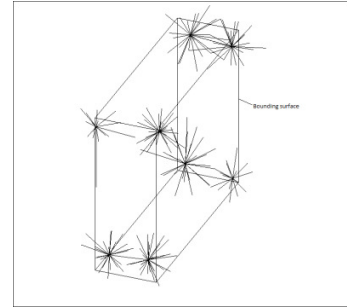
The combined cloud generation technique was tested for a generic wing-pylon-finned store case, for various store positions (shown in fig-4 and described in table-1). For each store position, combined cloud was generated from individual component cloud following the above mentioned technique. The resulting single large combined cloud was input to the NAL-MCIR mesh-less solver. The analysis were made for a flow mach number of 1.2 and an angle of attack of 0° .

RESULTS AND DISCUSSIONS

The combined cloud generation technique was successfully carried out and the solver ran seamlessly with the combined cloud as the input. Computational results show a good comparison with experiments as far as pressure coefficient (c_p) across the store body is considered. The pressure distribution in fin region, side force and roll moment computed numerically shows a slight mismatch with experiments, which can be eliminated by increasing cloud point density near fin region in the initial component cloud and modeling the boom (model support system used in experiment) in computations. Some typical c_p comparisons are shown in fig-5, 6 and 7.



a) Connectivity for points outside bounding surface



b) Connectivity for points inside bounding surface

Fig. 3. Point and its connectivity near the bounding surface

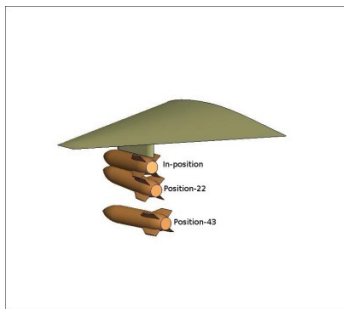


Fig. 4. Store positions for which CFD analyses were made

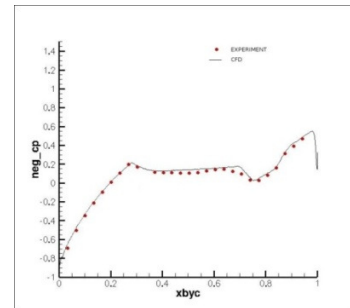


Fig. 5. C_p Comparison along store body at angle of 185° from pylon centre line for position-43

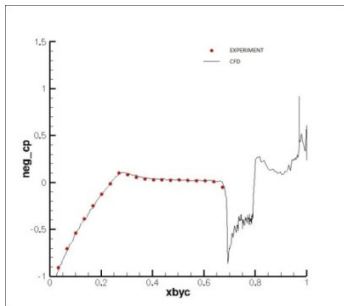


Fig. 6. C_p Comparison along store body at an angle of 45° from pylon centre line for position-43

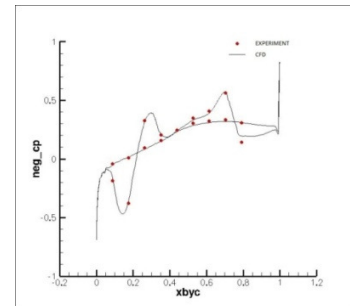


Fig. 7. C_p Comparison for wing at 45% span for position-43

POSITION	ROLL (o)	PITCH (o)	YAW (o)	X-TRANSLATION (1 unit = wing root chord)	Y-TRANSLATION (1 unit = wing root chord)	Z-TRANSLATION (1 unit = wing root chord)	COMBINED CLOUD SIZE (POINTS)
CARRIAGE- POSITION	0	0	0	0	0	0	1548144
POSITION-22	1.66	-5.18	-3.10	0.0074	-0.0031	0.0672	1362853
POSITION-43	6.51	-1.09	-9.41	0.0399	-0.0037	0.2020	1376839

Table-1. Details of store positions for which CFD analysis were made

ACKNOWLEDGEMENT

The present work has been carried out under a sponsored project from Boeing Company, USA (Purchase Contract No. 258396, Purchase Contract Change No: 02). Authors would like to thank Dr. Mori Mani, Boeing, for his support and advice during the course of the work. Authors would also like to thank Prof. N. Balakrishnan and team from Indian Institute of Science, Bangalore, India, for their suggestions and involvement in various stages of this work.

REFERENCES

- [1] Prewitt, N.C., Belk, D., and Shyy, W., "Multiple Body Trajectory Calculations Using the Beggar Code," AIAA paper 99-0913, 1999.
- [2] Scott M. Murman, Michael J. Aftosmis, Marsha J. Berger, "Simulations of 6-DOF Motion with a Cartesian Method," AIAA paper 2003-1246, 2003.
- [3] Praveen, C., "Development and Application of Kinetic Meshless Methods for Euler Equations," Ph.D. Thesis, Indian Institute of Science, Bangalore, India, 2004.
- [4] S.M. Deshpande, V. Ramesh, "Unsteady flow computations for flow past multiple moving Unsteady flow computations for flow past multiple moving," Computers & Fluids, ELSEVIER, Vol. 36, 2007.
- [5] A. K. Ghosh, S. M. Deshpande, "Least Squares Kinetic Upwind Method for Inviscid," AIAA paper 95-1735, 1995