

Interpolation Method needed for Numerical Uncertainty Analysis of Computational Fluid Dynamics

Curtis E. Groves¹ and Marcel Ilie, Ph.D.²
University of Central Florida, Orlando, FL 32816

Paul A. Schallhorn, Ph. D.³
National Aeronautics and Space Administration, Kennedy Space Center, FL, 32899

Extended Abstract

Using Computational Fluid Dynamics (CFD) to predict a flow field is an approximation to the exact problem and uncertainties exist. There is a method to approximate the errors in CFD via Richardson's Extrapolation. This method is based off of progressive grid refinement. To estimate the errors, the analyst must interpolate between at least three grids. This paper describes a study to find an appropriate interpolation scheme that can be used in Richardson's extrapolation or other uncertainty method to approximate errors.

Nomenclature

a	= channel width
dp/dx	= pressure gradient
ρ	= density
μ	= viscosity
\bar{V}	= average velocity
y	= distance from wall

I. Introduction

CFD is used in many forums to approximate flow solutions of the Navier-Stokes equations. The Navier-Stokes equations are second order, non-homogenous, non-linear partial differential equations. Several papers have been published on the use of progressive grid refinement to estimate the errors in a CFD Simulation^{1,2,3,4,5,6,7,8,9,10,11,12}. The procedure is to compare the differences in the solutions between at least three different grids. The computational domain discretization (grid) is significantly different in terms of the number of cells. This requires an

¹ Fluids Analysts, NASA Launch Services Program, VA-H3 & PhD Candidate, Department of Mechanical, Materials & Aerospace Engineering, University of Central Florida, AIAA Member.

² Assistant Professor, Department of Mechanical, Materials & Aerospace Engineering, AIAA Member

³ Environments and Launch Approval Branch Chief, NASA Launch Services Program, VA-H3, and AIAA Member.

interpolation between the grids and solutions to approximate the error. This interpolation will induce errors and the extrapolated uncertainty estimates become unreasonable and inaccurate ¹². The commercially available code ANSYS FLUENT includes mesh-to-mesh interpolation functionality ¹³. This method performs a zeroth-order interpolation (nearest neighbor) for interpolating the solution data from one mesh to another ¹³. This functionality is used to initialize data from one mesh on to another mesh for the purpose of an initial condition only. Using this method to approximate errors is inappropriate. OPENFOAM is an open source solver, which includes a similar functionality to the FLUENT mesh-to-mesh interpolation using a “mapfields” function ¹⁴. The mapfields function is also designed as an initial guess to be used when iterating a solution. Using the FLUENT interpolation file or OpenFOAM mapfields to estimate errors in a grid convergence study will produce unrealistic results. The purpose of this paper is to compare several other interpolation schemes that could be used for post-processing different solutions on different grids for the purpose of uncertainty estimation.

II. Interpolation Schemes

Matlab is a high-level language used for numerical computations and includes several interpolation functions for one-dimensional data, uniformly spaced, gridded data in two and three dimensions, and scattered data interpolation ¹⁵. CFD data comes in various forms, 1D, 2D, 3D, uniform, and non-uniform data. Matlab offers interp1, interp2, and interp3 for the corresponding dimensions. Interp1,2,3 includes the following schemes as shown in Table 1.

Table 1 – Interpolation Schemes available in Matlab

Interpolation Method	Matlab Function		
	interp1	interp2	interp3
'nearest' - Nearest neighbor interpolation	X	X	X
'linear' - Linear interpolation (default)	X	X	X
'spline' - Cubic spline interpolation	X	X	X
'pchip' - Piecewise cubic Hermite interpolation	X		
'cubic'	X	X (uniformly-spaced only)	X (uniformly-spaced only)
'v5cubic' - cubic interpolation used in Matlab 5	X		

A generic scheme is sought that would be available for 1D, 2D, 3D, uniform, and non-uniform grids. The three schemes available are “nearest”, “linear”, and “spline”. The nearest is the same scheme available in the current CFD

codes for mesh-to-mesh interpolation and would not provide additional benefit. The linear scheme and spline however could provide a better interpolation method for estimating numerical uncertainty in grid refinement studies.

III. Example Problem “Flow Between Parallel Plates”

Flow between parallel plates has an exact solution and provides a good example of the interpolation errors that can be induced by using the “nearest” scheme and will provide a metric for comparing the errors in the “linear” and “spline” to an exact solution.

Fully developed laminar flow between stationary, parallel plates is an exact solution to the Navier-Stokes Equations as derived in “Introduction to Fluid Mechanics”¹⁶. The width of the channel is (a).

$$u = \frac{a^2}{2\mu} \left(\frac{\partial P}{\partial x} \right) \left[\left(\frac{y}{a} \right)^2 - \left(\frac{y}{a} \right) \right] \tag{1}$$

A CFD model of this problem was created in FLUENT. The fluid is air. Table 2 outlines the parameters used.

Table 2 – Parameters

a (m)	0.1
rho (kg/m ³)	1.225
mu (Ns/m ²)	0.00001789
dp/dx (N/m ³)	-0.004

The exact solution is shown in Figure 1.

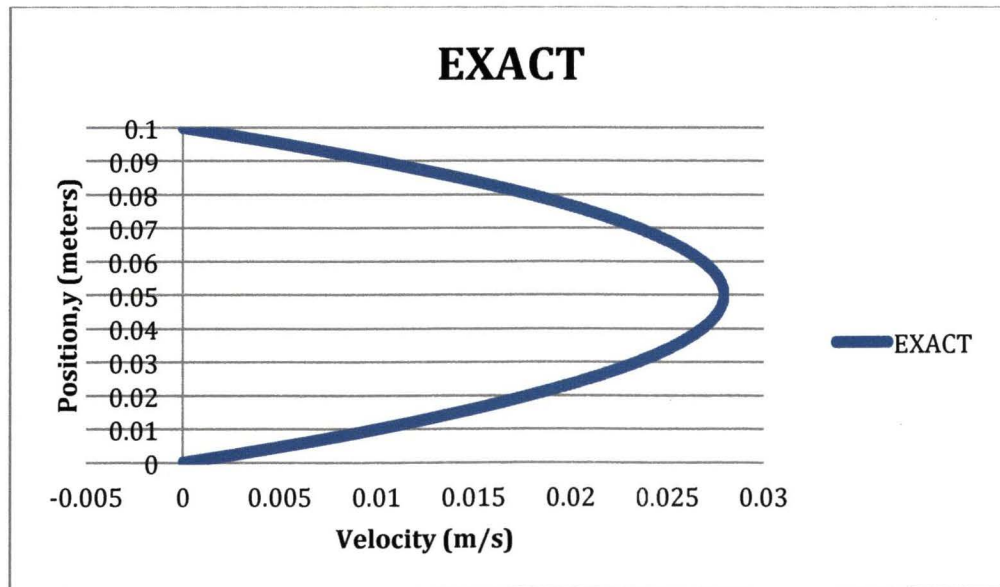


Figure 1 – Exact Solution

A CFD model was created for half of the domain. Flow between parallel plates has symmetry about the centerline. The inlet boundary condition used was the average velocity as shown in equation 2 and the domain was made long enough to be considered fully developed.

$$\bar{V} = -\frac{1}{12\mu} \left(\frac{\partial P}{\partial x} \right) a^2 \quad (2)$$

Three grids can be used to extrapolate an error. Three separate CFD models were created (coarse, medium, and fine). The coarse medium and fine grids have the following number of cells, 7140, 14186, 24780, respectively. The three solutions plotted for flow between parallel plates and compared to the exact solution are shown in figure 2.

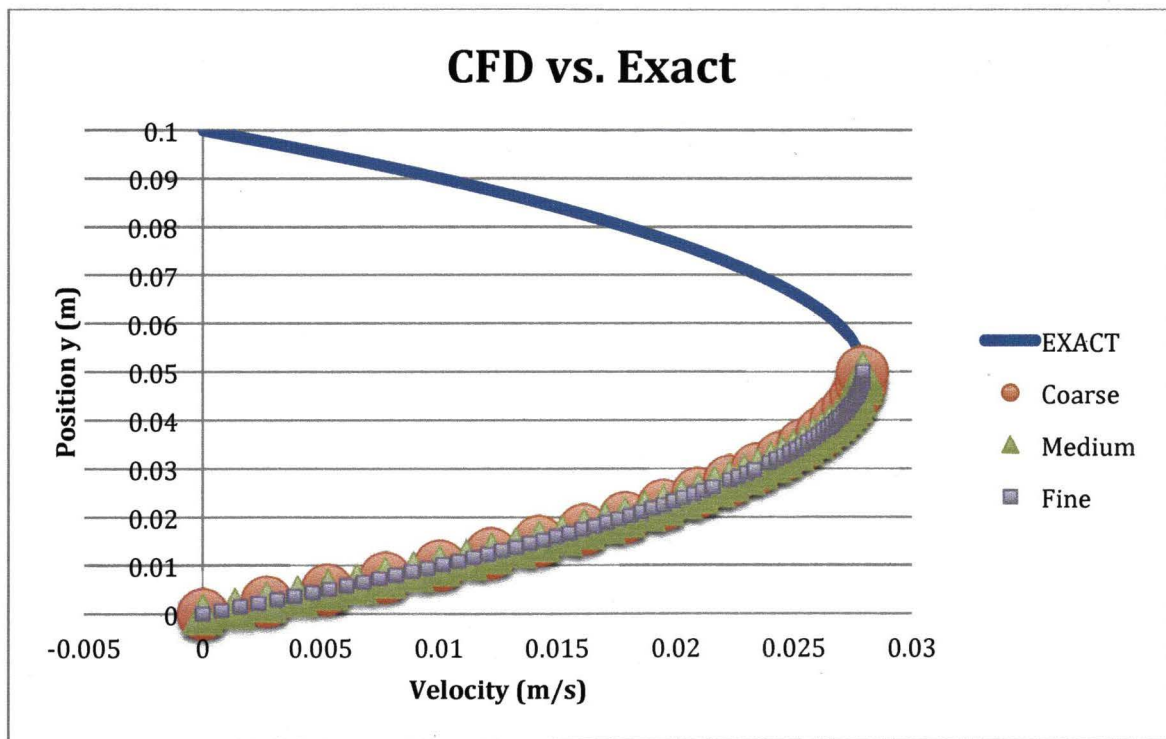


Figure 2 – CFD Results (coarse, medium, fine) vs. Exact Solution

IV. Results

The interpolation methods outlined “nearest”, “linear”, and “cubic” were investigated by interpolating the results from the fine grid and medium grid onto the coarse grid. The coarse grid was chosen because ideally it should be

good enough to approximate the solution and all recommendations for normal grid refinement already followed.

The medium and fine grids are used only for the error approximation.

FLUENT's mesh-to-mesh interpolation functionality was used and the results are shown in Figure 3. From a plot of the entire computational domain, the reader would not be able to see the variation, so the plot was zoomed in to show the errors being induced by using the "nearest" interpolation.

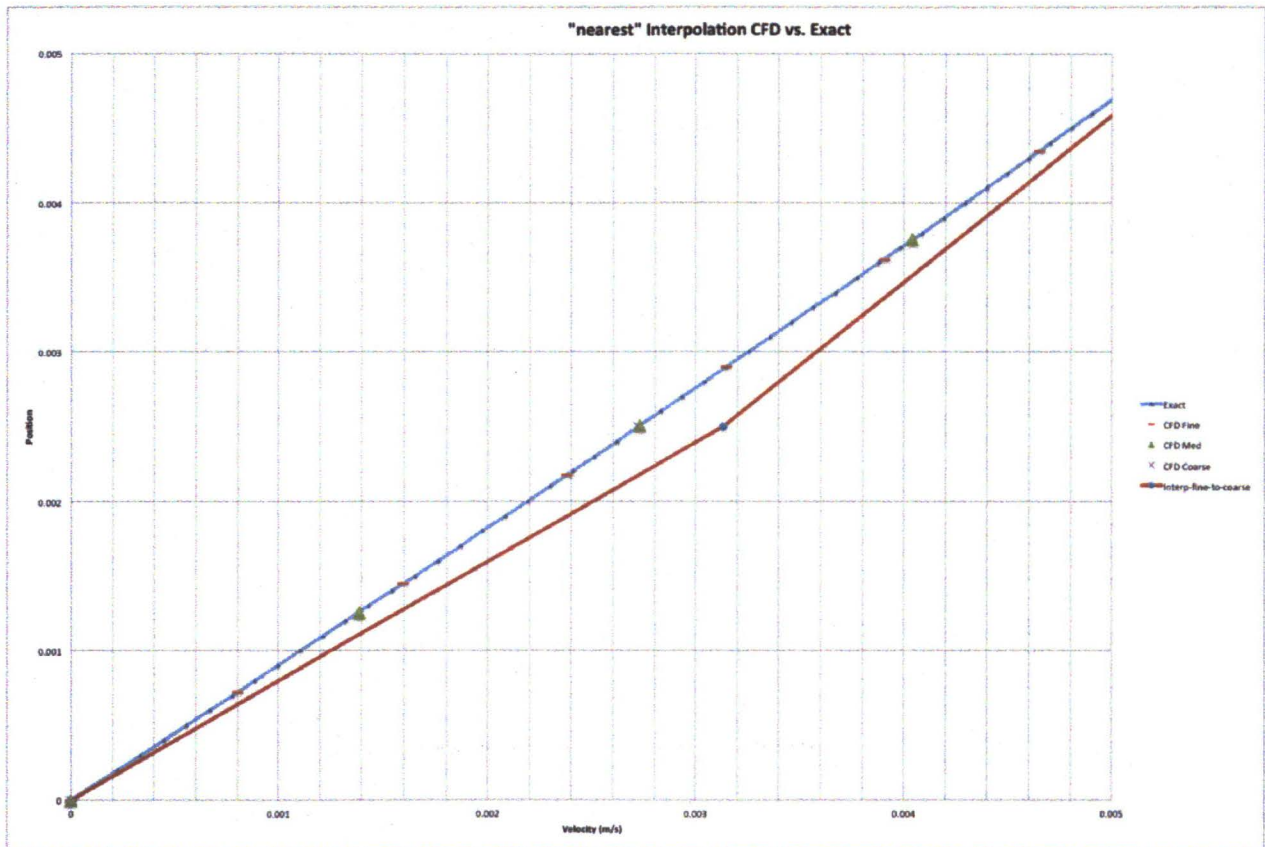


Figure 3 – "nearest" interpolation of CFD Results (coarse, medium, fine) vs. Exact Solution

The error induced by using the "nearest" interpolation scheme was as high as 15 percent of the exact value. Also, this high error was in the critical region closest to the wall.

To compare the linear interpolation scheme Matlab was used.

```
yfi = interp1(fine(:,2),fine(:,1),coarse(:,2),'linear')
```

The percent difference was greatly reduced to 0.08 percent of the exact solution and plotting the results does not visually show a difference as shown in figure 4.

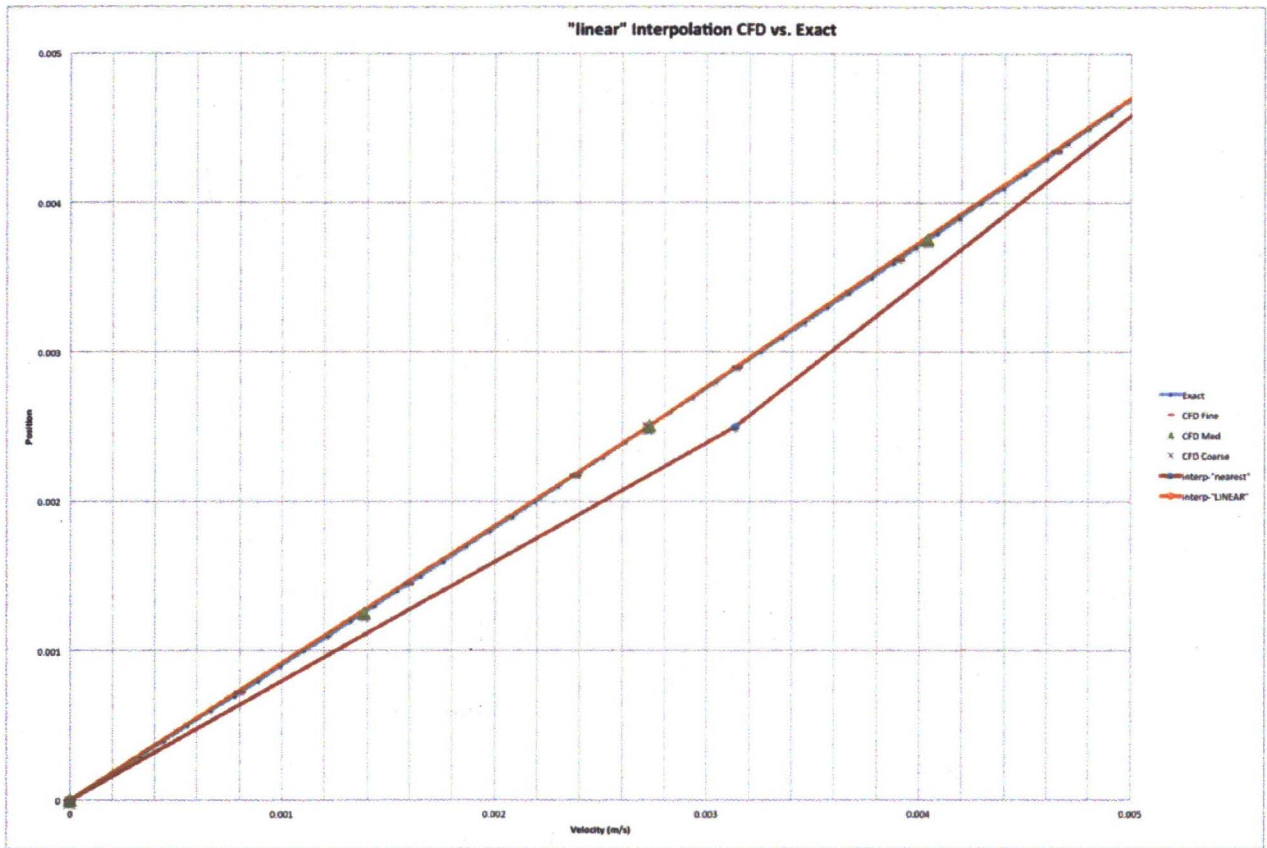


Figure 4 – "linear" interpolation of CFD Results (coarse, medium, fine) vs. Exact Solution

To compare the cubic interpolation scheme, Matlab was again used as follows.

```
yfi = interp1(fine(:,2),fine(:,1),coarse(:,2),'cubic')
```

The percent difference was even further reduced to 0.07 percent of the exact solution and plotting the results does not visually show a difference as shown in Figure 5.

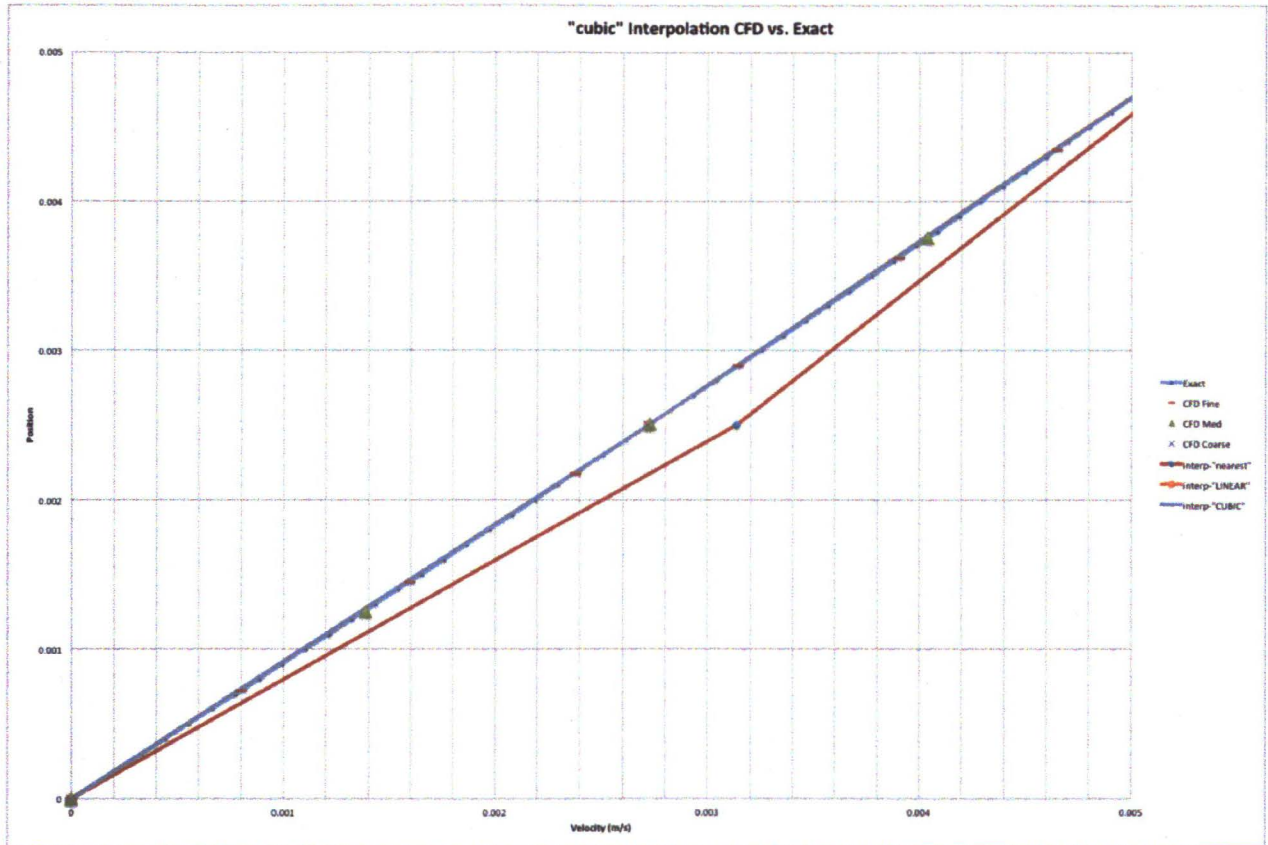


Figure 5 – “cubic” interpolation of CFD Results (coarse, medium, fine) vs. Exact Solution

The method will be extended to 2D and 3D to find the best interpolation method used for extrapolating errors between grids and included in the final manuscript.

V. Conclusion

A conclusion will be written comparing the interpolation schemes in one, two, and three dimensions and a method recommended to be used in CFD uncertainty and error estimation. The preliminary results show a “linear” method to be adequate, but the “cubic” to be the most exact for a one-dimensional problem.

References

- [1] Stern, F., Wilson, R. V., Coleman, H. W., and Paterson, E. G., "Verification and Validation of CFD Simulations," Iowa Institute of Hydraulic Research Report No. 407, September 1999.
- [2] Iudicello, F., "Guide for the verification and validation of computational fluid dynamics simulations," AIAA G-077-1998, 1998.
- [3] Celik, I., Karatekin, O., "Numerical Experiments on Application of Richardson Extrapolation With Nonuniform Grids," ASME Journal of Fluids Engineering, 119, Issue 3, p584.
- [4] Stern, F., Wilson, R. V., Coleman, H. W., and Paterson, E. G., 2001, "Comprehensive Approach to Verification and Validation of CFD Simulations - Part 1: Methodology and Procedures," ASME Journal of Fluids Engineering, 123, pp. 793-802.
- [5] Roache, P.J., "Verification and Validation in Computational Science and Engineering," Hermosa Publishers, 1998.
- [6] Celik, I., and Zhang, W., "Calculation of Numerical Uncertainty Using Richardson Extrapolation: Application to Some Simple Turbulent Flow Calculations," American Society of Mechanical Engineers Journal of Fluid Mechanics, 1995.

[7] Stern, F., Wilson, R. V., Coleman, H. W., and Paterson, E. G., "Verification and Validation of CFD Simulations," Iowa Institute of Hydraulic Research Report No. 407, September 1999.

[8] Stern, F., Wilson, R. V., Coleman, H. W., and Paterson, E. G., 2001, "Comprehensive Approach to Verification and Validation of CFD Simulations - Part 2: Application for Rans Simulation of a Cargo/Container Ship," ASME Journal of Fluids Engineering, 123, pp. 803-810.

[9] "Uncertainty Analysis in CFD Verification and Validation Methodology and Procedures," International Towing Tank Conference – Recommended Procedures and Guidelines, September 2008.

[10] Celik, I., Ghia. U., Roache, P., Freitas, C., Coleman, H., Raad, P., "Procedure for Estimating and Reporting of Uncertainty Due to Discretization in CFD Applications," American Society of Mechanical Engineers Journal of Fluids Engineering, Vol. 130, July 2008.

[11] An American National Standard., "Standard for Verification and Validation in Computational Fluid Dynamics and Heat Transfer", The American Society of Mechanical Engineers. ASME V&V 20-2009.

[12] Groves, C., Ilie, M., Schallhorn, Paul. "Comprehensive Approach to Verification and Validation of CFD Simulations Applied to Backward Facing Step," AIAA-2013-0258, 2013.

[13] "ANSYS FLUENT 12.0 User's Guide" Section 4.19 Mesh-to-Mesh Solution Interpolation

[14] "OpenFOAM User Guide" The OpenFOAM Foundation. <<http://www.openfoam.org/docs/user/>>

[15] Mathworks Product Website "Matlab" <<http://www.mathworks.com/products/matlab/>>

[16] Fox, Robert., McDonald, Alan, "Introduction to Fluid Mechanics" 3rd Edition. John Wiley & Sons, 1985