An Accurate Fast Fluid Dynamics Model for Data Center Applications

Wei Tian, Jim VanGilder, Michael Condor
Schneider Electric
800 Federal St.
Andover, MA, 01080
wei.tian@se.com, jim.vangilder@se.com, michael.condor@se.com

Xu Han, Wangda Zuo
University of Colorado Boulder
Boulder, Colorado, 80301
xu.han-2@colorado.edu, wangda.zuo@colorado.edu

ABSTRACT

Traditional CFD is broadly useful in the design and operation of reliable and efficient data centers. It is nevertheless computationally expensive, particularly when employed for design optimization, which usually requires multiple simulations. To speed-up calculations while retaining essential airflow physics, several researchers have turned to an alternative CFD methodology, namely, fast fluid dynamics (FFD). FFD has been reported to be much faster than traditional CFD, but at an assumed (acceptable) trade-off of reduced accuracy. However, a recent comparison of FFD and traditional CFD for data center plenum applications produced nearly indistinguishable results and suggested that previously-reported FFD/traditional-CFD differences were due primarily to inconsistent: 1) advection schemes, 2) computational grids, and 3) turbulence models. The present paper extends this FFD/traditional-CFD comparison to the data center whitespace and confirms the finding that a true like-for-like comparison produces nearly identical predictions.

Our FFD implementation utilizes a first-order upwind finite-volume scheme for advection like traditional CFD, so alternative advection schemes (e.g., semi-Lagrangian) are not considered further here. Likewise, the effect of grid choice on traditional-CFD predictions is well known. However, the effect of turbulence model for data center applications has not been reported extensively so we do consider this topic further here. We compare the standard k-ϵ and a simpler algebraic model to benchmark experimental data from a real data center. We find that the algebraic turbulence model predicts rack-inlet temperatures at a similar level-of-accuracy as the k-ϵ model for our fairly-simple-airflow reference data center.

KEY WORDS: CFD, Data Center, FFD, Fractional-Step, SIMPLE, Turbulence Model

NOMENCLATURE

T	temperature
ΔT	temperature difference
N	number of racks
$RMSE$	root mean square root

Greek symbols

ν	kinematic viscosity

Subscripts

i	index of the rack
t	turbulence
ref	reference value

Superscripts

FFD	prediction by FFD
CFD	prediction by traditional CFD
sim	simulated data
mea	measured data

INTRODUCTION

Data centers house mission-critical IT equipment which effectively dissipates all input power as heat during operation. As a primary heat removal method, air cooling typically features a raised-floor plenum to distribute the airflow from the cooling equipment to the racks. CFD has been widely employed to model the airflow and temperatures in data centers [1][2][3]. A recent study showed that, with careful calibration, traditional CFD can make reliable predictions of perforated-tile-airflow rates and rack-inlet temperatures [4].

CFD can also be employed to optimize designs and control cooling airflow in data centers, for example [1][5]; however, these applications typically require that multiple CFD simulations run in rapid succession. With traditional-CFD simulations requiring minutes to hours, researchers have naturally looked for faster alternatives.

One faster alternative has commonly been referred to as fast fluid dynamics (FFD); it solves the same Navier-Stokes equations as traditional CFD but utilizes an inherently-transient fractional-step method [6] instead of the more common semi-implicit method for pressure-linked equations (SIMPLE) [7] for pressure-velocity coupling. The name FFD consistently includes the fractional-step method but the approach taken to model advection varies. Some researchers have employed a semi-Lagrangian method [8] and various interpolation-accuracy orders [9] have been reported. Tian et al. [10] employed a first-order upwind finite-volume advection scheme like most traditional CFD, citing improved accuracy with only modest compromise on speed.

The speed benefits of FFD have been widely reported. FFD has been reported to be inherently 15-50 times faster than traditional CFD to arrive at steady-state solutions [8][11]. For purely transient simulations, the speed-up can be greater still. Further, FFD is more-easily parallelized than traditional CFD; running FFD on a graphics processing unit (GPU) has provided an additional factor of 30-1000 speed-up [12][13]. This level of speed improvement makes FFD a promising tool for design optimization and control applications.

However, the reported accuracy of FFD has been mixed. Zuo and Chen [8] described FFD as an "intermediate" approach between nodal models and traditional CFD, and found FFD was less accurate than traditional CFD. Dorostkar and Wang [14] reported a similar conclusion. Jin et al. [11], however, reported FFD predicted results more accurate than traditional CFD when compared to experimental benchmarks. Note that most of the above studies focused on solution speed while attempting to preserve a meaningful level of accuracy. Consequently, coarse grids and simpler turbulence (or laminar) models were generally selected. Tian et al. [10] compared FFD and traditional CFD in two applications: a simple hypothetical plenum with a strong recirculation pattern and the plenum of the real reference data center discussed in the present paper. They showed that, when identical advection schemes, computational grids, and turbulence models were utilized, the predictions of FFD and CFD were nearly indistinguishable from one another. In other words, the accuracy difference due to the fractional-step and SIMPLE methods appeared to be negligible, at least, for some data center plenum applications. The present paper extends the work of Tian et al. [10] to the data center whitespace and confirms the finding of nearly-indistinguishable accuracy.

Inconsistent computational grid and turbulence models have been implicated as factors leading to reported differences between FFD and traditional CFD. In such comparisons, it is relatively easy to ensure identical grid counts and distributions. For general advice about grid size for data center applications, see [15]. As far as the effect of turbulence model is concerned, little has been reported for data center whitespace, so we are left to refer to studies for general rooms [16][17], large electrical enclosures [18], and the data-center-plenum-only analysis of Tian et al. [10]. The general-room studies typically point to a variation of the k-ϵ model as the best choice. In smaller, more-confined geometries like large electrical enclosures, the choice of turbulence model does not make much difference. In [10] , a hypothetical plenum with an exaggerated recirculation revealed meaningful differences between algebraic and the standard k-ϵ model. In the present paper, we also compare the standard k-ϵ model to a simpler algebraic model and find that, for the reference data center considered here, with its fairly simple airflow patterns, both models deliver a comparable level-of-accuracy relative to measured data.

Although additional work – beyond that cited and presented here – is required, it is reasonable to predict that FFD with the first-order upwind finite-volume advection scheme will prove to be of essentially equal accuracy to traditional SIMPLE-based CFD for data center and, likely, similar applications such as general room ventilation, and electronics cooling. FFD has already shown to be dramatically faster than traditional CFD, particularly for transient applications. While anecdotally, FFD appears to be at least as stable as or more stable than traditional CFD, more work is required in this area.

Finally, on the question of turbulence model choice for data-center applications, the present work shows little difference between the k-ϵ and an algebraic model for the whitespace considered. However, there appears to be enough of difference in applications with strong recirculations that, for now, the k-ϵ should probably be favored over simpler algebraic models, at least, for plenum applications.

DESCRIPTION OF REFERENCE DATA CENTER

The work following in this paper is divided into two parts: 1) the comparison of FFD and traditional-CFD predictions, and 2) the comparison of turbulence models – both in the context of the data center whitespace. Our reference data center for these comparisons is a 7,400 ft^2 (690 m^2), raised-floor facility located in Massachusetts. In the first part, FFD and traditional-CFD results are simply compared to one another. In the second part, simulations are compared to experimental data which was gathered during an audit of the data center in March 2018.

The schematic of the data center is shown in Figure 1. It is approximately 100 ft (30.5 m) long and 74 ft (22.6 m) wide with a dropped-ceiling approximately 11 ft (3.4 m) above the raised floor. Other dimensions can be estimated from the figure noting that floor tiles are 2 ft (0.6 m) square. The total power consumption by the 151 racks and 12 PDUs is approximately 344 kW. All IT racks have a capacity of 42U except for the 18 racks located in Rows 1 and 10 which are 45U networking racks. There are 183 25%-open-area perforated tiles which receive cooling airflow through a 2 ft (0.6 m) plenum fed by two central air handling units; cooling airflow enters along the vertical sides of the plenum as shown in Figure 1. The total cooling airflow rate is 89,200 cfm (152,000 m^3/hr). Warm air is drawn into a ceiling through perforated ceiling tiles located above the hot aisles and ultimately returns to the central air-handling units.

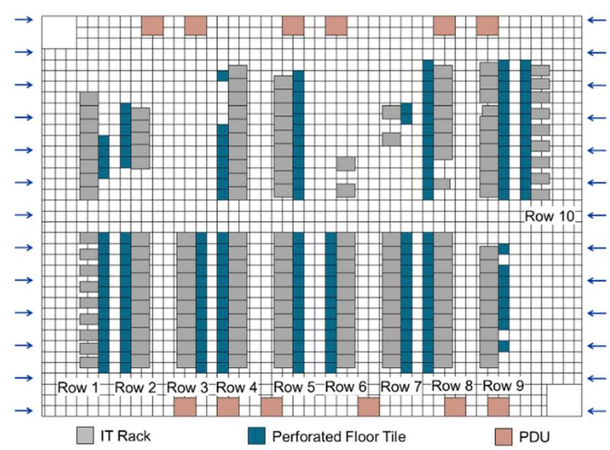


Fig. 1 Schematic of the Reference Data Center

The on-site measurements included rack-inlet temperatures which are used in the present study. Four individual rack inlet temperatures were measured at different heights along the centerline of each rack inlet. Measurements were made in accordance with the description provided in [4].

COMPARISON OF FFD AND TRADITIONAL-CFD PREDICTIONS

As discussed above, the comparison described in [10], showed that the FFD and traditional-CFD predictions were nearly identical when advection schemes, computational grids, and turbulence models were identical. Under these conditions, the remaining difference and ultimate point of comparison becomes the methods employed by FFD and traditional CFD for pressure-velocity coupling. Pressure-velocity coupling, in turn, refers to the strategy used to compute the (coupled) three velocity components and pressure. FFD utilizes a fractional-step method while traditional CFD most often utilizes SIMPLE. Both techniques have been widely documented in the literature, e.g., [19] and [7]; however, we provide a basic overview here.

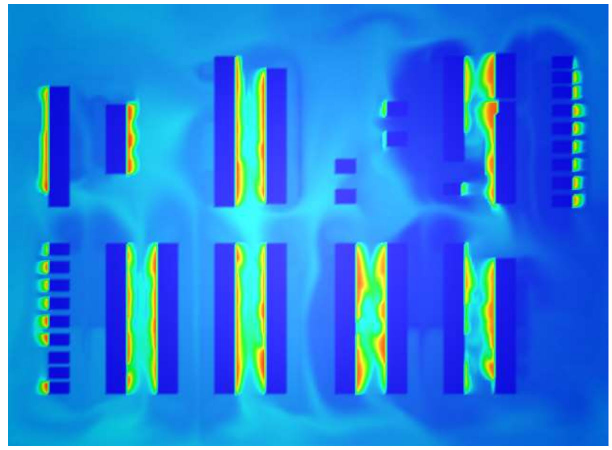
The fractional-step method is inherently transient and allows the momentum equations to be broken up into smaller pieces which may be solved separately. For example, the effects of advection and diffusion can be solved separately, in different fractions of a single time step. The very last fractional step before completing a full time step is called “projection”; in this fractional step, pressure is determined such that the continuity equation is satisfied. Note that our implementation of FFD includes all terms in the momentum equation (except pressure) in a single fractional step. This is different from the implementations presented in [8] and [11], which isolate advection so that a semi-Lagrangian scheme [20] can be applied. Our FFD implementation also utilizes a first-order upwind finite-volume discretization scheme to spatially discretize the momentum and energy equations – identical to that utilized by our reference traditional-CFD code, FloVENT [21].

SIMPLE does not need to be transient; steady-state solutions may be determined directly. It employs a pressure-correction scheme which involves an initial guess for pressure and subsequently a “pressure correction” after solving momentum equations to get an improved estimate. Iteration is required to determine the pressure and velocity fields which satisfy both momentum and mass-conservation equations.

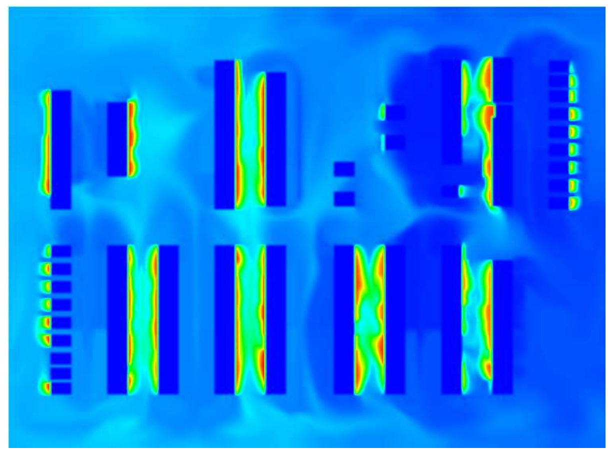
In both FFD and traditional CFD, the Boussinesq approximation was employed for density variations due to temperature differences. Uniform, 6 in (0.15 m) grid distributions were specified in both tools which resulted in 651,200 total cells. We used a simple constant-viscosity turbulence model that could be easily specified in both tools; the turbulent viscosity is fixed at 300 times the molecular kinematic viscosity ($\nu_t = 300\,\nu = 0.00405\ \mathrm{s/m^2}$). The traditional-CFD simulation was performed as “steady state” while the inherently-transient FFD simulation was run until steady-state was achieved using 0.1-second time steps.

Boundary conditions were specified identically in both FFD and traditional CFD. Only the space between the raised floor and the dropped ceiling was modeled. The perforated floor and ceiling tiles were modeled as simple uniform-velocity boundaries and racks were modelled as simple black-boxes with airflow distributed uniformly over the entire inlet and exhaust faces; the vertical temperature gradient was carried through from the inlet to the exhaust as recommended in [22]. The airflow was assumed to be proportional to total rack power dissipation with 125 cfm (212 $\mathrm{m^3/hr}$) of airflow for each kW of power. PDUs were modeled simply as adiabatic blocks.

Figure 2 shows the predicted temperature contours at 5.5 ft (1.7 m) above the raised floor - which intersects the rack at approximately 85% of its height. FFD and traditional CFD predicted the temperature distributions that are virtually indistinguishable despite the fact that the traditional-CFD plot was created in its native interface while an open-source post-processing tool was used for FFD plot. In any case, the overwhelming agreement between the models is the same as previously observed with the floor-plenum application [10].



a) FFD



b) Traditional CFD

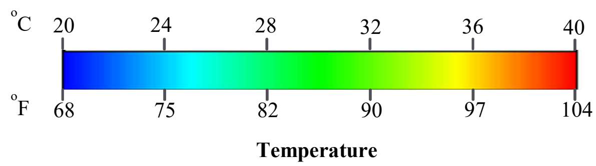


Fig. 2 Temperature Contours at Mid-Room Height

To quantify the differences between FFD and traditional CFD, we compared the average of temperatures taken at four positions along the height of the rack inlets (consistent with experimental measurements). The percentage difference was computed as $100 \times \left|T_i^{FFD} - T_i^{CFD}\right| / \Delta T_{ref}$ where T_i^{FFD} and T_i^{CFD} are the temperatures determined by FFD and traditional CFD, respectively; the reference temperature range, ΔT_{ref}, was taken as 25°F (14°C), which corresponds to the temperature rise across the racks. Note that a relative difference of 0.2% corresponds to an absolute difference of 0.05°F (0.03°C). Figure 3 shows the number of racks which fall into a given percentage-difference range. For example, for 138 racks, the difference between FFD and traditional CFD is in the range from 0.0-0.2%. The maximum difference, recorded for one rack, is approximately 2.5% which corresponds to an absolute difference of 0.6°F (0.4°C).

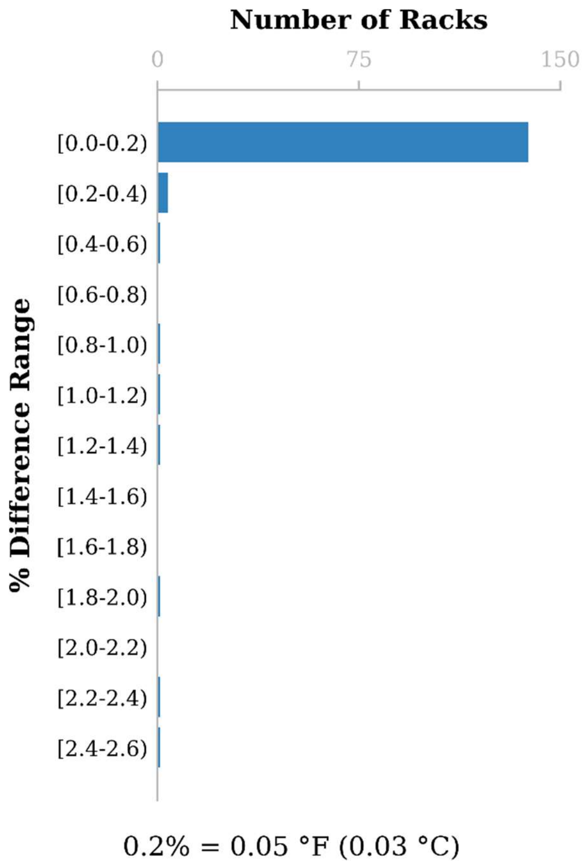


Fig. 3 Average Rack-Inlet Temperature Differences Between FFD and Traditional CFD

Finally, the root-mean-square-error ($RMSE$) that quantifies the average difference over all racks was computed as:

$$RMSE = \sqrt{\frac{\sum_{i=1}^{N}\left(\frac{T_i^{FFD} - T_i^{CFD}}{\Delta T_{ref}}\right)^2}{N}} \tag{1}$$

where N is number of racks (or inlet-temperature predictions). The overall $RMSE$ is 0.4%, which indicates that the typical difference is approximately 0.1°F (0.06 °C) in absolute terms. Considering these results together with the temperature contours of Figure 2, we conclude that the fractional-step method in our FFD implementation delivers results that are nearly indistinguishable from those of SIMPLE in traditional CFD, at least, for this reference data center. Before leaving this section, we report the solution times observed for FFD and traditional CFD for this like-for-like data-center-room simulation. The traditional CFD simulation was performed on a single core Intel Xeon central processing unit (CPU) E3-1220 v6 on a Windows 10 workstation with 32 GB of random-access memory. The FFD simulation was performed on an NVIDIA Quadro P2000 GPU on the same workstation. The traditional-CFD simulation took 1 hour 5 minutes and 40 seconds to converge (2000 iterations) while FFD required 6 minutes and 20 seconds. FFD was approximately 10.4 times faster than traditional CFD. We note that this data does not constitute a rigorous benchmarking of solution speed as it is possible to run the traditional CFD solution, at least, on multiple cores, if not a GPU. Further, this is a comparison of commercial CFD to an in-house FFD implementation so it is impossible to control for differences in how the tools were written and compiled. Finally, convergence parameters, which are very case-dependent, may or may not be particularly optimized for the current application in either code.

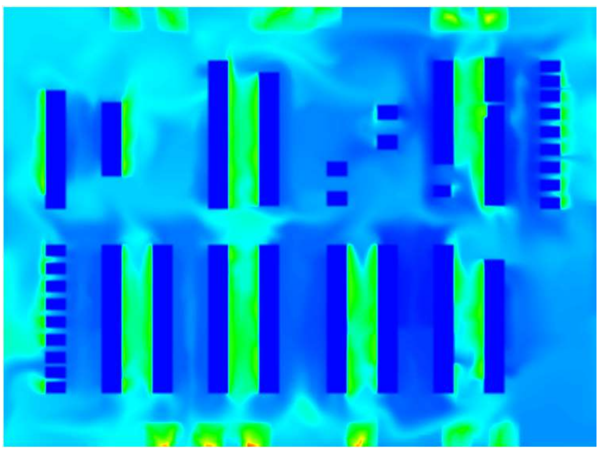
COMPARISON OF TURBULENCE MODELS

In the previous section, we performed a comparison between (in-house) FFD and traditional (commercial) CFD for the reference data center whitespace. For convenience in achieving a true like-for-like comparison, we used a model that was slightly simplified. In the present section, we now aim to compare turbulence models for whitespace modeling – both relative to one another and experimental measurements. Seeking reasonable absolute accuracy, we added a couple of details to the whitespace models. First, to more accurately model the “jet effects” associated with the perforated tile airflow, we added a momentum source to the first grid cell above each perforated tile as suggested by Abdelmaksoud et al [23]. Second, we added prescribed surface heat fluxes to the PDUs (based on measured values) where they were previously modeled as adiabatic. Finally, we chose to compare practical turbulence models and, therefore, no longer considered the overly-simplistic constant-viscosity model.

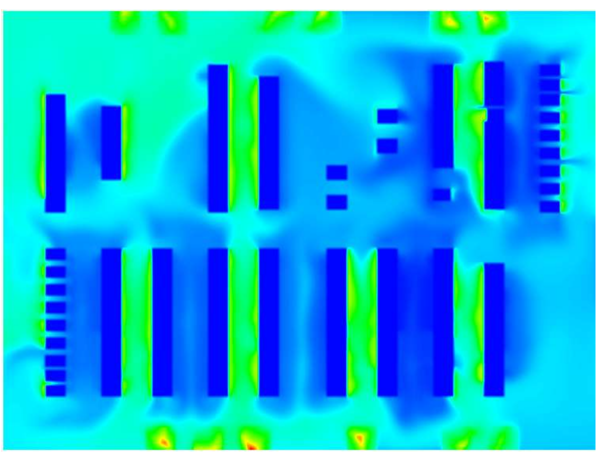
As there has been little research in the literature on turbulence modeling specific to data-center applications, the k-ϵ (two-equation) model [24] is typically the default choice – as it is for many other applications. Chen compared a few variants of k-ϵ models in simulating basic indoor-airflows and ranked the RNG k-ϵ model as the top performer among all tested models [17]. Chen and Xu [25] proposed an algebraic (zero-equation) turbulence model for indoor airflow modeling and concluded that it can predict indoor airflow patterns and temperatures with reasonable accuracy. The LVEL algebraic model was proposed by Agonafer et al. [26] especially for electronics-cooling applications. Dhoot et al. [18] proposed a revised algebraic model that incorporates an approximate-wall function into the Chen and Xu model, which improved simulation accuracy for large-electronics-enclosure applications. Note that the algebraic models require less computation per iteration than the k-ϵ model; however, it is too simplistic to declare them faster overall because, the turbulence model affects the rate of convergence. Further, in the algebraic models, the required calculation of a characteristic length scale between the fluid-cell center adds to computational cost. For example, the LVEL model described in [26] relies on the solution of an additional Poisson equation prior to the CFD simulation to estimate the length scale for all fluid cells. Exact methods for determining the length scale can also be employed [18] which also require upfront computational cost.

To carry out a like-for-like comparison, we chose the LVEL algebraic model [26] and the standard k-ϵ (2-equation) model [24] primarily because both are conveniently available in our traditional CFD benchmark tool [21]. We note that the LVEL simulation did not quite reach full convergence; however, all residuals dropped to approximately "10" using the default criteria on a logarithmic scale on which "1" is considered full convergence.

Figure 4 shows the temperature contours at 5.5 ft (1.7 m) above the raised floor - at approximately 85% of the rack height for the two turbulence models. Cold-aisle predictions, where the flow patterns are simple and heavily influenced by prescribed-velocity boundary conditions, are all very similar. Overall, the algebraic model predicted obviously-lower room temperatures than the standard k-ϵ model which would intuitively seem to generate more mixing.



a) LVEL algebraic model [26]



b) Standard k-ϵ model [24]

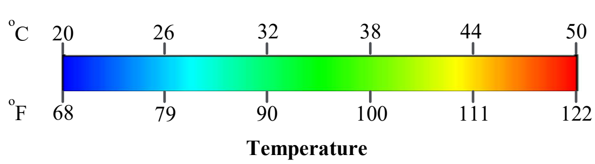


Fig. 4 Temperature Contours at Mid-Room Height

To further quantify the comparisons and reveal absolute accuracy relative to experimental measurements, we computed the percentage difference of the average rack-inlet temperatures as $100 \times \left|T_i^{sim} - T_i^{mea}\right| / \Delta T_{ref}$, where T_i^{sim} and T_i^{mea} are the simulated and measured values, respectively. Figure 5 shows the number of racks falling into a given percentage-difference range (relative to measured data) for the two turbulence models considered. The prediction accuracy is very similar for the two models. The fact that the distribution of predictions relative to experimental data follows such a similar pattern indicates that the differences between CFD models and experiment, in general, are larger than those between turbulence models. Consequently, the main conclusion here may be that other potential sources of error in the models dominate and turbulence model is not the primary concern.

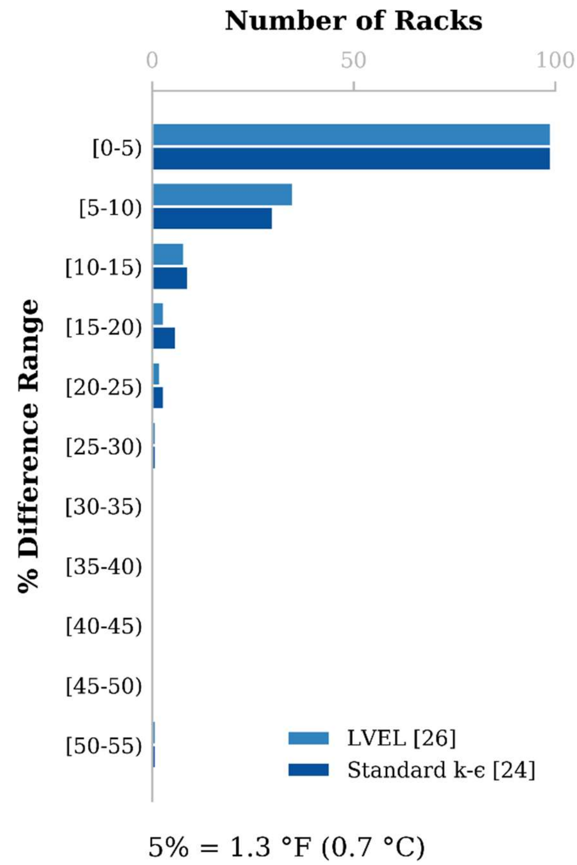


Fig. 5 Average Rack-Inlet Temperature Differences Between Turbulence Model and Experimental Measurements

RMSE can once again be utilized to assess the overall difference between simulated and measured temperatures. These values are 7.8%, and 8.4% for LVEL and k-ϵ models, respectively. Like the more detailed distributions shown in Figure 5, these values are sufficiently close to one another so as to further suggest that turbulence model choice is not the primary driver of differences between predicted and measured values.

Though not the focus of the current work, likely sources of error relative to measured data include: 1) transient variations in data-center-supply temperature occurring over the several hours required to measure rack-inlet temperatures, 2) inaccurate rack-by-rack power which had to be crudely estimated based on the power consumption of large groups of racks, 3) inaccurate rack airflow values which were assumed proportional to (already inaccurate) rack-power values, and 4) overly-simplistic black-box rack models which did not provide airflow leakage paths through and under racks.

CONCLUSIONS

FFD, which utilizes a factional-step method has been shown to be substantially faster than traditional CFD, which typically utilizes a SIMPLE-based method for pressure-velocity coupling. Consequently, FFD is a promising candidate for design-optimization and data-center-control applications which often require multiple simulations. While previous research indicated that the faster speed of FFD was achieved with a compromise in accuracy, we recently showed in [10] that our FFD solver produced results nearly indistinguishable to those of traditional CFD for a hypothetical data-center plenum application. In [10], we utilized a first-order upwind finite-volume advection scheme and ensured that grid and turbulence models were implemented consistently in both simulations.

The present paper first extends the work of [10] by modeling the airflows and temperatures in the whitespace of a reference data center. Again, FFD and traditional-CFD results are nearly indistinguishable from one another. Though not a rigorously like-for-like comparison, we note that FFD running on a modest GPU reached the steady-state solution approximately 10-times faster than the commercial CFD running on a single-CPU core.

We then extended the turbulence-modeling investigation for plenums of [10] to the data-center whitespace. In this case, we find that the LVEL algebraic model [26] achieved essentially the same level of accuracy as the widely-used standard k-ϵ model for predicting rack-inlet temperatures in our reference data center. However, it is noted that the airflow patterns in the data center whitespace were fairly simple and absent of strong recirculations.

Considering the present work and prior studies, we conclude that FFD with a first-order upwind finite-volume (i.e., the same as typically employed in traditional CFD) advection scheme can be expected to provide essentially the same accuracy as traditional CFD for data center applications (assuming other modeling choices like computational grid and turbulence model are identical). So, FFD has now been shown to be fast and accurate. Its stability, is yet to be quantified but we anecdotally report that it is as robust as or more robust than traditional CFD in convergence.

On the question of turbulence modeling, for now, we continue to recommend the widely popular k-ϵ model as its computational overhead is low while producing confidence-inspiring results over a range of data center applications including plenums and rooms.

ACKNOWLEDGMENTS

This research was supported in part by the U.S. Department of Energy under Contract No. DE-EE0007688 and the National Science Foundation under Award No. IIS-1802017.

The authors thank David W. Plamondon at University of Massachusetts Medical School for facilitating on-site data-center measurements.DISCLAIMER

DISCLAIMER

REFERENCES

[1] L. Marshall, and P. Bemis, "Using CFD for Data Center Design and Analysis", Applied Math Modeling White Paper, 2011.

[2] J. VanGilder, and R. Schmidt, "Airflow Uniformity through Perforated Tiles in a Raised-Floor Data Center", Proceedings of ASME 2005 Pacific Rim Technical Conference and Exhibition on Integration and Packaging of MEMS, NEMS, and Electronic Systems collocated with the ASME 2005 Heat Transfer Summer Conference, San Francisco, California, July 17–22, 2005.

[3] W. Abdelmaksoud, H. Khalifa, T. Dang, R. Schmidt, and M. Iyengar, "Improved CFD Modeling of a Small Data Center Test Cell", Proceedings of Thermal and Thermomechanical Phenomena in Electronic Systems (ITherm), Las Vegas, Nevada, June 2-5, 2010.

[4] Z. Pardey, J. VanGilder, C. Healey, and D. Plamondon, "Creating a Calibrated CFD Model of a Midsize Data Center", Proceedings of InterPACK, San Francisco, California, July 6-9, 2015.

[5] L. Billet, C. Healey, J. VanGilder, and Z. Pardey, "Data Center Cooling Efficiency with Simulation-Based Optimization", Proceedings of ASME 2015 International Technical Conference and Exhibition on Packaging and Integration of Electronic and Photonic Microsystems collocated with the ASME 2015 13th International Conference on Nanochannels, Microchannels, and Minichannels, San Francisco, California, July 6–9, 2015.

[6] J. Ferziger, and M. Peric, *Computational Methods for Fluid Dynamics*, Springer, 2002.

[7] S. Patankar, *Numerical heat transfer and fluid flow*, CRC press, 1980.

[8] W. Zuo, and Q. Chen, "Real-Time or Faster-than-Real-Time Simulation of Airflow in Buildings", Indoor air, vol. 19, no. 1, pp. 33–44, 2009.

[9] M. Mortezazadeh, and L.Wang, "A High-Order Backward Forward Sweep Interpolating Algorithm for Semi-Lagrangian Method", International Journal for Numerical Methods in Fluids, vol. 84, no. 10, pp. 584–597, 2017.

[10] W. Tian, J. VanGilder, X. Han, C. Healey, M. Condor, and W. Zuo, "A New Fast Fluid Dynamics Model for Data-Center Floor Plenums", Proceedings of 2019 ASHRAE Winter Conference, Atlanta, Georgia, January 12-16, 2019.
[11] M. Jin, W. Zuo, and Q. Chen, "Improvements of Fast Fluid Dynamics for Simulating Air Flow in Buildings", Numerical Heat Transfer, Part B: Fundamentals, vol. 62, no. 6, pp. 419–438, 2012.
[12] W. Zuo, and Q. Chen, "Fast and Informative Flow Simulations in a Building by Using Fast Fluid Dynamics Model on Graphics Processing Unit", Building and Environment, vol. 45, no. 3, pp. 747–757, 2010.
[13] W. Tian, T. Sevilla, and W. Zuo, "A Systematic Evaluation of Accelerating Indoor Airflow Simulations Using Cross-Platform Parallel Computing", Journal of Building Performance Simulation, vol. 10, no. 3, pp. 243–255, 2017.
[14] M. Dorostkar, and L. Wang, "A Fast and Accurate CFD Solver for Indoor Airflows", Proceedings of eSIM, Hamilton, Ontario, May 3-6, 2016.
[15] J. VanGilder, and X. Zhang, "Coarse-Grid CFD: The Effect of Grid Size on Data Center Modeling", ASHRAE Transactions, vol. 114, no. 2, pp. 166–182, 2008.
[16] Q. Chen, and W. Xu, "A Zero-Equation Turbulence Model for Indoor Airflow Simulation", Energy and Buildings, 1998.
[17] Q. Chen, "Comparison of Different K-ε Models for Indoor Air Flow Computations", Numerical Heat Transfer, Part B: Fundamentals, 1995.
[18] P. Dhoot, J. VanGilder, Z. Pardey, and C. Healey, "Zero-Equation Turbulence Models for Large Electrical and Electronics Enclosure Applications", Proceedings of 2017 ASHRAE Winter Conference,Las Vegas, Nevada, January 28-February 1, 2017.
[19] J. Stam, "Stable Fluids", Proceedings of the 26th annual conference on Computer graphics and interactive techniques, Los Angeles, California, August 08-13,1999.
[20] R. Courant, E. Isaacson, and M. Rees, "On the Solution of Nonlinear Hyperbolic Differential Equations by Finite Differences", Communications on Pure and Applied Mathematics, vol. 5, no. 3, pp. 243–255, 1952.
[21] Mentor, "FloVENT 12.1", www.mentor.com, 2018.
[22] J. Zhai, K. Hermansen, and S. Al-Saadi, "The Development of Simplified Rack Boundary Conditions for Numerical Data Center Models", ASHRAE Transactions, vol. 118, p. 436, 2012.
[23] W. Abdelmaksoud, H. Khalifa, T. Dang, B. Elhadidi, R. Schmidt, and M. Iyengar, "Experimental and Computational Study of Perforated Floor Tile in Data Centers", Proceedings of Thermal and Thermomechanical Phenomena in Electronic Systems (ITherm), Las Vegas, Nevada, June 2-5, 2010.
[24] B. Launder, and D. Spalding, "The Numerical Computation of Turbulent Flows", Computer Methods in Applied Mechanics and Engineering, 1974.
[25] J. VanGilder, C. Healey, Z. Pardey, and X. Zhang, "A Compact Server Model for Transient Data Center Simulations", ASHRAE Transactions, vol. 119, p. 358, 2013.
[26] D. Agonafer, L. Gan-Li, and D. Spalding, "LVEL Turbulence Model for Conjugate Heat Transfer at Low Reynolds Numbers", American Society of Mechanical Engineers, EEP, vol. 18, pp. 23–26, 1996.