Djunaedy, E., Hensen, J., & Loomans, M. (2004).
Building energy simulation and computational fluid dynamics: 1+1>2?.
Proceedings of the TI - IBPSA-NVL studiedag, October 21, pp. 15. Antwerpen: Technologisch Instituut KVIV.

Gebouwsimulatie en Computational Fluid Dynamics (CFD): 1+1>2?

E. Djunaedy, J. Hensen and M. Loomans

Center for Building and Systems TU/e – TNO (http://www.kcbs.nl)
email: M.Loomans@bouw.tno.nl

Abstract

De toepassing van een numerieke simulatie dient vraaggestuurd te zijn. Een keuze voor een bepaalde simulatie moet in principe niet worden gemaakt omdat het zo een interessant programma is of omdat het zulke mooie plaatjes oplevert.

De paper beschrijft de toepassing van verschillende wijzen van simulatie van de luchtstroming in een ruimte en een gebouw. Het meest belangrijke onderwerp hierbij is de weloverwogen keuze die gemaakt moet worden voordat een bepaald programma of een bepaalde aanpak gebruikt wordt. Ieder programma heeft immers zijn mogelijkheden en beperkingen. Om deze keuze te begeleiden wordt een procedure beschreven die dit proces kan ondersteunen. In de procedure wordt koppeling van verschillende programma´s als optie meegenomen om te komen tot betere resultaten.

In het tweede gedeelte van de paper wordt vervolgens ingegaan op deze koppeling, meer specifiek op de koppeling van gebouwsimulatieprogramma's en stromingssimulaties (CFD). Er wordt een overzicht gegeven van de randvoorwaarden en de mogelijkheden. Tot slot worden resultaten van een case study gepresenteerd die de potentie van koppeling van gebouwsimulatie en stromingssimulatie aantonen, en dan met name ook van externe koppeling.

De essentie van de paper is echter dat voor een ontwerpvraag, indien op basis van een objectieve beoordeling gewenst is dat gekoppelde (stromings)simulatie wordt toegepast, "1+1" meer kan zijn dan "2".

Deze paper is een compilatie van drie papers die eerder zijn gepubliceerd door de auteurs (Djuneady et al. 2003; Djuneady et al. 2004; Djuneady et al. 2005) en is gebaseerd op lopend promotieonderzoek van Ery Djunaedy bij het Center for Building and Systems TU/e – TNO. Voor meer achtergrondinformatie over het onderwerp wordt naar deze papers verwezen.

Building energy simulation and Computational Fluid Dynamics: 1+1>2?

E. Djunaedy, J. Hensen and M. Loomans

Center for Building and Systems TU/e – TNO (http://www.kcbs.nl)
email: M.Loomans@bouw.tno.nl

1 Introduction

Building simulation is rapidly becoming the most important engineering tool in integrated design of buildings and HVAC systems. Recent advances in hardware and software make simulation – that required the use of supercomputers in the past – widely available, even to a small consulting firm with a single PC.

Also different approaches have become available. In terms of resolution and complexity, currently indoor airflow modeling can be categorized into three approaches:

1. Building energy balance models (BES) that basically rely on guessed or estimated values of airflow.
2. Zonal airflow network (AFN) models that are based on (macroscopic) zone mass balance and inter-zone flow-pressure relationships; typically for a whole building.
3. Computational Fluid Dynamics (CFD) that is based on mass, momentum and energy conservation in all (minuscule) cells that make up the flow domain; typically a single building zone.

The practical availability and easy access to different simulation methods and tools poses the question of how to justify the selection of a certain method or tool.

Furthermore, every simulation program has its own limitations. For CFD, the boundary conditions of CFD are usually assumed with limited consideration for the thermal storage effects of the wall, external conditions and interactions with building services systems. For BES, the energy prediction is calculated based on a well-mixed assumption so that the definition of the convective heat transfer coefficient (CHTC) cannot capture the dynamics of the flow near the surfaces.

Therefore, as a fourth approach the coupling of building energy simulatioon (BES) and computational fluid dynamics (CFD) simulation has been discussed in various publications in recent years, e.g. Negrao (1998), Srebric et al (2000), Beausoleil-Morrison (2002), Zhai et al (2002). Coupling of BES and CFD has the potential to generate better results because the two can provide boundary conditions to each other. For example, BES can provide internal surface temperatures of the walls to CFD, while CFD can provide more accurate CHTC for BES. The benefits of the coupled simulation has been discussed in the publications mentioned.

This paper first discusses a procedure for a more objective choice for an indoor airflow simulation resolution program. Then the implementation of (exteral) coupling between BES and CFD is described and an example result is presented to justify the use of the coupling approach, and to highlight its potentials, or put otherwise, to indicate whether 1 + 1 can be more than 2. This paper is a compilation of three papers that have been written earlier by the authors (Djuneady et al. 2003; Djuneady et al. 2004; Djuneady et al. 2005) and is based on a ongoing PhD research by Ery Djunaedy at the Center for Building and Systems TU/e – TNO. For more in depth information reference is made to these papers.

2 Capabilities and applicability

Hensen et al (1996) analyzes the capabilities and applicability of the various approaches described above in the context of a displacement ventilation system. One of the main conclusions of this work was that a higher resolution approach does not necessarily cover all the design questions that may be answered by a lower resolution approach. Each approach has its own merits and drawbacks. An environmental engineer typically needs each approach but at different moments in the design process. The main conclusions of this study are summarized in Table 1.

Table 1. Summary of prediction potential (--=none, ++=very good) for airflow simulation resolution levels in case of displacement ventilation (Hensen et. al. 1996)

Aspect	BES	AFN	CFD
Cooling electricity	--	++	--
Fan capacity	++	++	--
Whole body thermal comfort	+	++	+
Local discomfort, gradient	--	+	++
Local discomfort, turbulence intensity	--	--	++
Ventilation efficiency	--	0	++
Contaminant distribution	-	-	++
Whole building integration	++	++	--
Integration over time	++	++	--

Each program has its own technical shortcomings. E.g., for CFD the domain boundary is usually the inside surface of a room. However, it is difficult to predict the corresponding boundary conditions (Beausolleil-Morrison, 2000) since these depend on many parameters and variables, e.g. construction details, ambient conditions and HVAC operation. In BES, the specification of convective heat transfer is simplified – and its importance often underestimated - by using surface averaged film heat transfer coefficients (h_c). Many studies (e.g. Spitler et. al. (1991) and Lomas (1996)) however show that the specification of h_c has significant impact on the result (e.g. up to 37% difference in energy consumption prediction).

Integration or coupling of CFD and BES is seen as an opportunity to improve the performance of the separate methods. The main questions however are what selection process should be used to justify the use of a certain (integrated/coupled) approach. For the selection process a methodology has been developed that guides the process of deciding whether and which method or coupling procedure should be applied.

3 Coupling Procedure Decision Methodology

3.1 Methodology

When various design options have to be assessed, it will be evident that a high resolution (and more complex) approach may require significantly more resources in terms of computing capacity, manpower and time. On the other hand lower resolution (and less complex) approaches may not reliably solve a particular problem. How to select the appropriate approach to solve the design problem at hand remains a challenge.

In practice – especially with inexperienced users – one may observe the tendency to use the most sophisticated and highest resolution method to simulate a design option. Obviously, the perceived sophistication of a high resolution approach would not fail to impress any client. However, the additional cost of using a higher resolution method should always be justified. Failing to do so will only risk that a client will hesitate to use the method – or simulation in general - in the future.

A prototype Coupling Procedure Decision Methodology (CPDM) has been developed to help the decision making on what approach should be used for a particular problem. The main ideas behind the CPDM as schematically shown in Figure 1 are:

1. A simulation should be consistent with its objective, i.e. the designer should not be tool-led (the use of tools – usually the most sophisticated – without considering the apropriateness of the tools for the problem at hand).
2. There should be a problem-led (not tool-led) rationale to progress from one level of resolution and complexity to the next.
3. The selection of a better design option (among many design options) should be made at the lowest resolution possible, so that less design options need to be simulated at a higher resolution level.

The vertical axis of the chart in Figure 1 represents layers of different resolution of building simulation. Increasing levels of resolution are, e.g. energy simulation, airflow network simulation, and CFD simulation.

Each of the resolution layers is separated by one or more decision layers. The horizontal axis shows the different levels of complexity of the building simulation.

The first step is to select the minimum resolution based on the design question at hand. For example:

- If energy consumption is of interest, then BES would probably be sufficient.
- If a temperature gradient should be predicted, then at least an AFN is required.
- If local mean age of air is in question, then CFD is necessary.

A second step is to check whether the above minimum resolution is actually sufficiently accurate for the design question at hand. For example:

- Load analysis may be over-sensitive to assumed values of infiltration rate or inter-zonal ventilation, thus requiring AFN to predict more accurate infiltration airflow rates.
- Load analysis based on BES may be over-sensitive to the convective heat transfer coefficient (h_c), thus requiring CFD to predict more accurate h_c values.

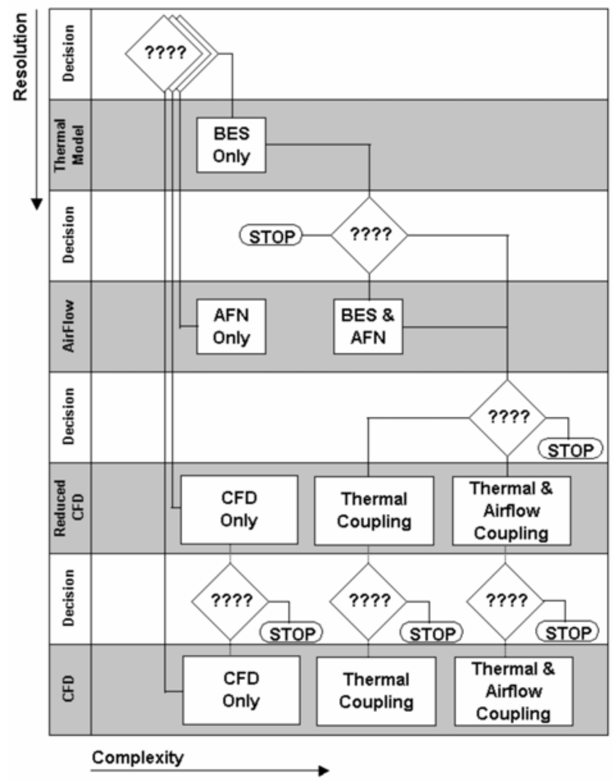


Figure 1. Prototype airflow based Coupling Procedure Decision Methodology.

Starting from the top of the chart, there should be an assessment whether a certain case should be simulated at the simplest level of complexity at a certain level of resolution. If the problem cannot be solved at the simplest level of complexity - whatever the resolution is - then a certain form of coupled simulation is required.

How to actually make these decisions is to a large extend still vague as denoted by the question marks in Figure 1. In practice the decisions are often made implicitly and they very much depend on the skills and experience of a design engineer. Instead, e.g., sensitivity analysis is proposed as a decision making tool. In this proces the decisions in building design are based on performance indicators. Table 2 presents some typical performance indicators (PI) that are of interest to a building engineer.

Table 2. Performance indicators and minimum required modeling resolution in terms of simulation approach.

Performance Indicators	Approach
Energy Related	
a. Heating energy demand	BES
b. Cooling energy demand	BES
c. Fan electricity	BES
d. Gas consumption	BES
e. Primary energy	BES
Load Related	
f. Max heating load	BES
g. Max cooling load	BES
Comfort Related	
h. PPD	BES
i. Max temperature in the zone	BES
j. Min temperature in the zone	BES
k. Over heating period	BES
l. Local discomfort, temp gradient	AFN
m. Local discomfort, turbulence intensity	CFD
n. Contaminant distribution	AFN
o. Ventilation efficiency	AFN
p. Local mean age of air	CFD

With regard to the CPDM, these indicators can be used to select the appropriate approach to simulate the problem at hand. Table 2 indicates that only a few indicators require an immediate jump to higher resolution approaches as AFN or CFD. Energy demand and maximum load, both for heating and cooling, may well be calculated from BES. Fan electricity, gas consumption and primary energy, normally, are more difficult to simulate, as the simulation must include explicit system simulation. Instead fan electricity consumption, gas consumption and primary energy can be calculated manually using a BES simulation result as the base.

PPD (Percentage People Dissatisfied) is used as an indicator for the general thermal comfort of occupants in the space. The maximum and minimum zone temperature and the overheating period are relevant as comfort indicators for naturally ventilated building. For air-conditioned buildings these indicators determine the capacity of the installations.

3.2 Performance Assessment

The performance assessment is a two-step procedure. In the first step of the assessment it should be clear that the performance indicators have been derived at the correct level of resolution. The second step then deals with the judgment of the actually derived value and the required performance.

To assess the validity of the applied resolution level, sensitivity analysis is used as the decision tool. Sensitivity analysis is the systemic investigation of the simulation response to either extreme values of the model's quantitative factors or to drastic changes in the model's qualitative factors (Kleijnen 1997). This analysis is used in many fields of engineering as a what-if analysis, and one example of the use of this method in building simulation is given by Lomas and Eppel (1992).

The main use of sensitivity analysis is to investigate the impact of a certain change in one (or more) input parameter on the output. Depending on the particular problem, the end result is usually to identify which input parameter has the most important impact on the output.

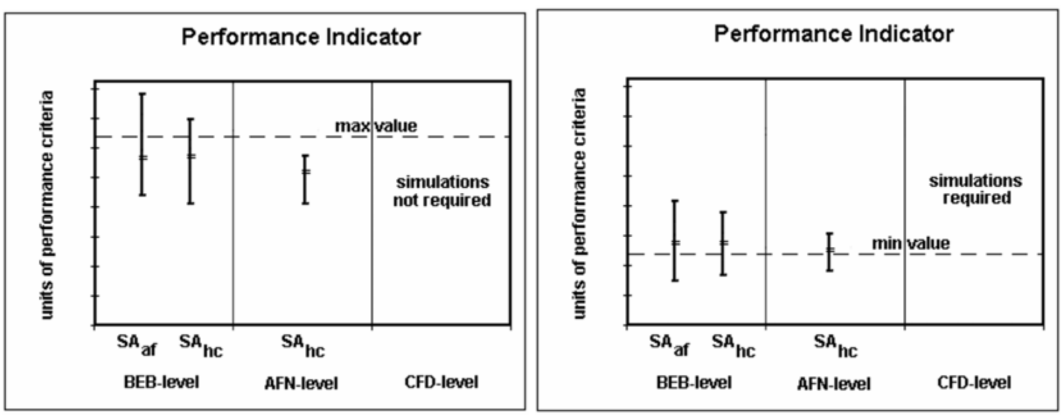
For the CPDM, the sensitivity analysis is used for a slightly different purpose. The sensitivity analysis is not used to identify which input parameter is important, but rather to identify the effect of changes in one input parameter to a number of outputs. Which input parameter is used in this study is selected from results of previous research in this area.

From previous studies, e.g. Hensen (1991), Negrao (1995), and Beusolleil-Morrison (2001), it is known that there are two main inputs that should be tested through a sensitivity analysis in order to decide if progress to a higher resolution level is necessary:

1. Airflow parameters assumption, especially the infiltration rate, for the decision to use AFN-coupled simulation.
2. Convective heat transfer coefficient, for the decision to use CFD.

For the "normal" use of sensitivity analysis where there are many input parameters and a single output, the effect of each input parameter can be quantified relatively easy. However, with the use of sensitivity analysis as described for the CPDM, it is intended to compare the effect of changes in one input parameter (e.g. airflow parameter) on different outputs (e.g. heating demand and heating load). Different outputs have different significance in different building designs. Therefore within the CPDM the performance indicators are not weighted. Instead, to quantify the effect of changes in input to the performance indicators, the difference in the performance indicator value when the input is changed from a minimum to a maximum value is used.

Figures 2 shows two possible scenarios for the sensitivity analysis. Here two examples are shown of results for two performance indicators with different target values. In Figure 2(a), a sensitivity analysis at the lower (BES-) resolution level indicates that the performance may not be met as a result of the large sensitivity for the convective heat transfer coefficient. The next level of resolution therefore is applied to calculate the specific performance. The sensitivity analysis for the (AFN-) resolution level indicates that the investigated design meets the required performance, also when including the sensitivity, and higher resolution simulation is not required for this performance indicator. The second example, Figure 2(b), shows that in another situation a further resolution increment may well be possible.



(a) Scenario 1 *(b) Scenario 2*

Figure 2. Assessment of the validity of the level of resolution through sensitivity analysis.

(S_{af}: sensitivity analysis on airflow parameters;) (S_{hc}: sensitivity analysis on CHTC)

The above methodology presents a possible procedure for applying different resolution approaches for building simulation (more specifically air flow modeling) in a more objective manner. The CPDM, furthermore, forms a framework for the integration of different approaches for air flow modeling. In the second part of this paper the focus will be shifted to this integration, in particular the integration of building energy simulation and airflow simulation.

4 Integration Of Building Airflow And Thermal Domain

4.1 Methods of integration

As described elsewhere in more detail (Djunaedy, 2002), there are two major categories on how to implement the integration of the building airflow and the thermal domain. The first method of integration is to include the dynamics fabric model and (optional) radiation model into CFD to form what is called the conjugate heat transfer method (e. g. Chen et. al. 1995 and Moser et. al. 1995). It is based on including the thermal characteristics and processes in the walls into the model so that the boundary condition of the CFD models can be moved from the inside to the outside of the wall, and thus capturing the dynamics of the external conditions.

This will result in a single set of equations to be solved in a single time step. All variables for both the fluid and the solid domains are up-dated simultaneously.

This is the direct answer to the problem of setting up boundary condition for CFD without considering the use of BES. In fact, there is a tendency to avoid the use of BES for at least two reasons (Chen et. al. 1995):

1. If the interior wall temperature is calculated by BES, then it needs the data (at least h_c) from CFD. There would be an iteration process between the two to get an agreement on the values, and this iteration is seen as inconvenient.
2. The conduction heat transfer calculation in most BES packages assumes one–dimensional heat transfer which will introduce errors.

However, the application of the conjugate heat transfer method has several disadvantages:

1. The difference in stiffness of the fluid and the solid side of the model will lead to difficulties in obtaining a converged solution (Chen et. al. 1995).
2. It is computationally expensive (Zhai et. al. 2001). The computing time increases dramatically because of the difference in the time scale between fluid (few seconds) and solid (few hours) so that the calculation must be performed over a long time path to include the dynamics in the solid, but over a very small time step to account for the dynamics in the fluid.
3. Most probably, the code of the solvers must completely be rewritten.

There have been few developments on this method, including a new algorithm to stabilize the computation process. However, Zhai et. al. (2001) conclude that this method is not practical for immediate use in the design context with current computer capabilities.

The second method of integration is to couple CFD and BES, where two separate simulation tools exchange data in a predefined way (e.g. Zhai et. al. 2001, Srebric et. al (2000), Beausolleil-Morrison et. al. (2001)). This method has some advantages that directly answer the restrictions of the previous approach:

1. There is no internal computational stiffness problem for either tool as the fluid and the solid side of the model are simulated separately.
2. It is computationally less expensive since it does not solve the whole equations at the same time, and thus there are no two different time scales to deal with.
3. The solvers for the separate domains can be optimized individually to account for the characteristics of the respective domains.
4. It is possible to use a separate program without rewriting the code. This is a major advantage as we can immediately use codes that are available from each domain, which have been developed separately over the years, well proven and benchmarked.

From the results of previous research, it can be concluded that coupled simulation is the most promising method of integration, between CFD and BES, for use in building design.

4.2 Coupled simulation

We define coupled simulation as two (or more) separate simulation tools, each of them solving a separate set of equations, that exchange time-step data between them in a prescribed manner. A coupled simulation usually involves the following components:

1. Domain solvers (It must be clear which code calculates which terms in the overall solution scheme).
2. Geometry modeller and/or grid generator.
3. Master program which coordinates the coupling procedure, e.g.:
   - Frequency and point in the solution procedure where data is exchanged between the codes.
   - Definition of the variables that will be passed between the codes.
   - Method of time step control

Based on the interaction between the domain solvers, coupled simulation can be categorized into:

1. *Internal coupling*, where the domain application is tailored to work specifically within a certain environment. Usually the code needs to be rewritten for this.

2. *External coupling*, where the domain application is not changed to cooperate with other domain applications (For this study, “external coupling” is defined as run-time communication between two separate programs where at least one of the programs continues to run while exchanging information with the other program.)

ESP-r (Clarke, 2001) is a good example to clarify this categorization. It uses internal coupling to couple the energy balance model with the airflow network (Hensen, 1991), and with CFD (Negrao (1995) and Beusolleil-Morrison (2000)). On the other hand it uses external coupling to couple the thermal balance model (and airflow models) with lighting application (Janak et. al., 2001).

In the current work we concentrate on external coupling for two main reasons:

1. Individual domain applications have evolved separately over the years and are well proven. If we know how to make these different domain applications to communicate with each other, then it would be of great advance to the building industry. Rewriting the code can be seen as a set back from these independent advances in the separate domains.
2. Each individual domain can be developed further independently. There is no need to worry about keeping up with the latest development in each domain. Let each domain expand and progress in their respective directions. As it is known how the domains can communicate with each other, it is possible to take advantage of these latest developments.

External coupling has also been adopted in other fields of science and technology, for example in structural aerodynamic, nuclear plant safety, car crash simulation, medicine, and earth climate.

4.3 Some Issues On External Coupling

The first issue is about the stability of the iteration between CFD and BES in order to obtain a converged value on the data being exchanged, as indicated by Chen, et. al. (1995). This issue has been addressed specifically in a recent study by Zhai et. al. (2003). He concluded that the solution of CFD-BES coupling does exist and is unique.

The second issue is about the “mechanics” of the external coupling. Three main aspects define the mechanics of the external coupling:

1. The definition of the variables that will be passed between the codes; The domain connection between CFD and BES is located on the inside wall of the room. There are several possibilities on which parameters are to be exchanged, e.g. convective heat transfer coefficient (h_c), temperature (T), heat flux (Q), and the airflow rate (V). It must be clear which application program calculates which parameters and that parameter definitions are the same.
2. The frequency and point in the solution procedure where the data is exchanged between the application programs; Should the data be exchanged every time step? Or every hundred time steps? For every exchange of data, should there be an iteration until convergence, or just exchange and march on to the next time step?
3. The method of time step control; Should both simulations be dynamic? If both are transient or quasi-steady-state, should they have the same time step?

The third issue is about the use of different geometry modelers. Citherlet et. al. (2001) indicate several disadvantages if we have to create more than one model for a single project. Firstly, it is time consuming. Secondly, any modification in the project has to be translated between models. However, this should be seen as a price for doing a coupled simulation. Furthermore, there are some developments in the area of application interoperability where the issue of data management of computer models is one of the main issues (Karola et. al. 2001). It is expected that this problem will be resolved if these developments mature.

4.4 Different implementations of coupling strategies

The focus point of coupling between CFD and BES can be represented by the convection heat transfer equation on the internal surfaces:

$$q_c = hA(T_{wall} - T_{ref}),$$

where q_c is the convective heat flux [W], h is the conventive heat transfer coefficient (CHTC) [W/m^{2}K], A is the wall surface area [m^2], T_{wall} is the wall temperature [°C] and T_{ref} is the reference temperature [°C]. Most of the available BES packages assume a well-mixed condition so that the room can be represented by a single temperature. CFD can overcome this limitation and determine the air temperature near the surfaces of the room.

The convection coefficient can then be calculated using these temperatures, and the result is sent back to BES to improve the calculation for the next time step.

For each CFD run, the surface temperatures are obtained from the BES. Furthermore, the heat extraction rate from BES can be used to determine the inlet boundary conditions in the CFD calculation. Since the heat flows and surface temperatures vary with time in buildings, it is necessary theoretically to run CFD for each time-step.

Based on the interaction between the two packages, Negrao (1995) proposed two coupling mechanisms: integrated coupling and surface coupling. In integrated coupling, CFD interacts directly with the thermal matrix solver and resolves the exchanged parameters until the values are converged. CFD is used to solve the zone air-point temperature and the internal surface convection, while the BES provides the CFD with the internal surface temperatures. In every time step, both iteratively exchange the data until convergence before moving to the next time step.

In surface coupling, on the other hand, the two programs work independently and exchange information at the internal surfaces. The CFD uses the boundary conditions (i.e. wall temperature, and supply air parameters if applicable) from the previous time step, calculates the convection heat transfer coefficient (CHTC) and sends this value back to the BES. The BES will then use this information to form the matrix for the zone heat balance equations and solves the matrix for the current time step. The simulation continues with CFD simulations always using the data from the previous time step.

Beausolleil-Morrison (2000) argued that surface coupling brings many advantages over the integrated approach. For external coupling, the most important feature is that the surface coupling provides more flexibility in defining the coupling mechanism. With regard to accuracy, obviously integrated coupling is more accurate because it resolves the exchanged data in many iterations until converged to a certain value. However, the accuracy will be the same if the time step is sufficiently small. In principle the integrated coupling mechanism can be implemented for external coupling once there is a faster mechanism for data exchange.

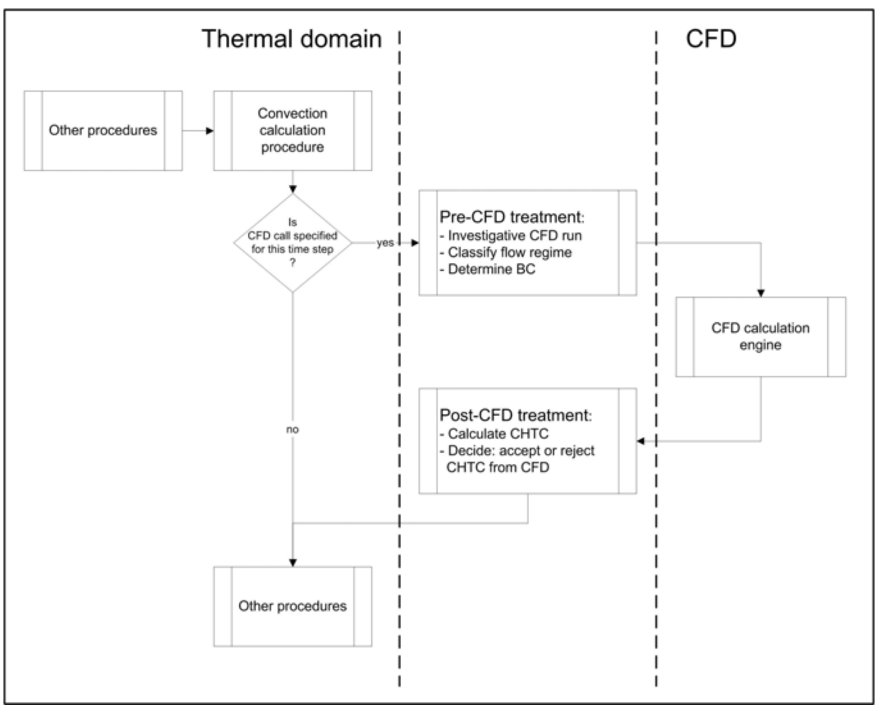


Figure 3. BES-CFD coupling.

Furthermore, the CFD-predicted value of CHTC ($CHTC_{CFD}$) can always be rejected in surface coupling. This cannot be done in integrated coupling without interrupting the iteration process. This checking mechanism of the $CHTC_{CFD}$ value before passing it back to BES is one of the quality assurance measures that should be used when using surface coupling.

Figure 3 gives an overview of the current status of surface coupling (Beausolleil-Morrison, 2000). In summary, for every time step during the calculation of the convection heat transfer coefficient (CHTC) of internal surfaces, the thermal domain checks whether there is any CFD call defined for that time step. If not, it continues with another mechanism for defining the CHTC of the internal surfaces. If yes, it will invoke the coupling controller to derive the CHTC from a CFD simulation. The coupling mechanism consists of a pre-CFD treatment, (final) the actual CFD simulation and a post-CFD treatment. Further detailed explanation can be found in reference (Beausolleil-Morrison, 2000).

4.5 Implementation of external coupling

As a starting point, an advanced building energy program (ESRU 2000) and a commercial CFD package (Fluent 2003) were selected. Even though the initial implementation and the validation studies were performed with these two programs, the results of this study can be applied to any BES and CFD.

For external coupling, BES (in this case ESP-r) does the calculation on the thermal domain (the left hand side of Figure 3), while CFD (in this case Fluent) does the calculation on the CFD domain. For the pre- and post-CFD treatment, a Unix script has been created to act as the coupling controller.

The zero equation turbulence model (Chen and Xu, 1998) was used for the CFD simulation because it uses less computing resources. Furthermore, it has been successfully used for the coupling of CFD and energy simulation (Chen et al. 1999).

The CHTC can be explicitly defined using the Reynolds analogy:

$$h = \frac{\mu_{eff}}{\mathrm{Pr}_{eff}} \frac{c_p}{\Delta x}$$

where h is the convective heat transfer coefficient [W/m^{2}K], μ_{eff} is the effective viscosity [kg/m.s], Pr_{eff} is the effective Prandtl number (=0.9) [-], c_p is the specific heat [J/kg.K] and Δx is the distance between the surface to the adjacent cell [m].

The following mechanism reflects the latest developments on external coupling between CFD and BES:

1. During the calculation of the CHTC for the internal surfaces BES checks if a CFD simulation is specified for the current time step. If not, it will continue to calculate the CHTC based on user input.
2. If yes, the BES will invoke the coupling controller which will call the external CFD program to run the CFD simulation.
3. After the simulation, the CFD program calculates the CHTC for each internal surfaces and sends the result back to the coupling controller.
4. If the CHTC falls within pre-defined criteria, the coupling controller will pass the result to the BES, otherwise the coupling controller will send a flag to BES to use its own CHTC value.

A more detailed description of the implementation of the external coupling method is described elsewhere (Djunaedy, 2005). An example of the application of coupled simulation will be described in the next paragraph. In this example also a comparison is made with internal coupling, to indicate its potential.

5 Case study

5.1 Case description

The performance of the internal and external coupling methods was assessed by simulating an experimental atrium (Figure 4). The atrium is located in Yokohama, Japan, and has been studied extensively to produce high quality data for the validation of simulations of large spaces, as part of a research program coordinated by IEA Annex 26 (Heiselberg et al., 1998). Many simulation studies have used these sets of data for validation purposes.

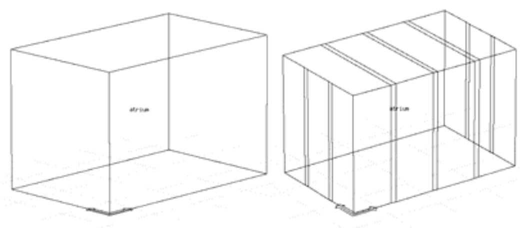
The atrium is facing south and has three glass walls (south, east, and west), a glass roof, and two insulated surfaces (north wall and floor). The full description of the atrium and the experimental settings can be found in references (Hiramatsu et al., 1996; Ozeki et al., 1996; Murakami et al., 1994).

The selected setting for this study was denoted in the reports as Case 2(a). For this case, the north wall and the floor were painted white. The measurement was carried out without any ventilation and air conditioning on a spring day (31-Mar-1994). The measurement setting, methodology and results are well-documented and can be found in reference (Murakami et al. 1994).

For this study, two models were created (Figures 5a and 5b). The first model is the simplified model of the atrium where the steel frames are left out. The second model includes the steel frames.

The steel frame is actually very complex to model. Apart from the external frame (which holds the glass), there is an internal frame which holds the overall structure of the atrium. The steel construction is estimated to have a volume of $0.46m^3$ and a total surface area of $72m^2$.

Figure 4. IEA Annex 26 Atrium (Yokohama, Japan)



(a) (b)

Figure 5. IEA Test Room geometry.

The geometry of the frames was not modeled as in reality, because it would lead to a very complex model for the CFD simulation. The total frame area from the outside surface is kept similar (around 10% of the glazed area). Furthermore, to capture the thermal capacity of the frame, the total volume is also kept similar to the reality. However the steel surface area on the inside surface is very underestimated.

Apart from the shape of the steel frame, the specification of the steel properties brings in another level of difficulty. The properties of the steel frame are not specified in the report. Furthermore, the standard steel properties as provided in the BES program could not obtain the correct temperature rise during the simulation compared to the measured value. Trial-and-error was used to obtain the correct number.

The report estimates the leakage from the glazed structure. However, the simulations were carried out with zero infiltration rates.

The simulations were carried out for 24 hours with a time-step of 15 minutes. A CFD simulation was invoked every time step, so in total there are 96 steady-state CFD simulations for every coupled case.

There were six cases studied, three cases for each geometrical model. The uncoupled cases were simulated using ESP-r without CFD coupling. The CHTC for this case was calculated using the Alamdari-Hammond correlation. These two cases are considered the base case. The internal coupling cases were simulated using ESP-r, coupled internally with the internal CFD module. The external coupling cases were simulated using ESP-r and Fluent. Table 3 shows the settings of the cases.

Table 3. Simulation parameters.

Model	CFD	Mesh	Time
Simple	No CFD	n.a.	0:01:31
Simple	Internal coupling	10x10x10	1:09:37
Simple	External coupling	35x23x23	4:08:58
With frame	No CFD	n.a.	0:02:59
With frame	Internal coupling	19x14x14	4:22:40
With frame	External coupling	35x23x23	5:53:30

5.2 Results and discussion

5.2.1 Computation

The CFD coupled simulation, both internal coupling and external coupling, used the zero equation model. Both also used the same convergence criteria (i.e. 10^{-6} residuals error for energy and 10^{-3} for others). The number of iterations for the CFD simulation was limited to 500 iterations, and the iteration always starts from the latest data from the previous time step.

The meshes of the internal coupling cases were intentionally made coarser due to the solver limitations. On the simple model, with a mesh of 10x10x10, the internal coupling case took 1 hr 15 min to run. When the mesh was refined, due to the geometrical demand to include the frame, to 19x14x14 (less then double), the computing time became more than 4 hrs (almost four times). The external coupling, on the other hand, can use a finer mesh (uniform grid size of 0.2m) with a more or less similar computing time.

In general, external coupling can have up to twice finer CFD mesh with comparable computing time. However, this conclusion is very much dependent on the choice of the solver. The point that we want to highlight is that external coupling offers a freedom to choose the most powerful solver without the need of much coding work.

5.2.2 Air temperature

Figures 6 and 7 show the comparison between the measurement and simulation results of the air temperature. The measured air temperature is actually an average value from 27 measurement points. For this measurement setting, such averaging is valid as the temperature can be considered uniform. As shown in Figures 6 and 7, coupled simulation can bring the temperature down to be closer to the measured value.

Two noticeable differences between the simulation and the measurement are the peak temperature prediction and the rise (and decay) of the temperature. The simulation predicts higher peak temperature compared to the measurement and the temperature prediction rises (and decays) in a faster rate compared to the measurement. These two differences are solely attributed to the input uncertainties in the material (thermal and optical) properties.

Both pictures also show the effect of explicitly modeling the frame, which brings down the temperature prediction closer the measured value. The temperature also rises (and decays) at different rates, due to the thermal capacity of the frame.

5.2.3 CHTC

Figure 8 shows the comparison of CHTC values of the south wall for the three cases of the simplified model. The fluctuation in the coupled simulation result (both internal and external coupling) actually shows the coupling controller in action.

In general, the CFD-coupled simulation will predict a higher value of CHTC because it does not use the well-mixed assumption. However, the coupling controller will check the CHTC value to ensure that it falls within an acceptable range (not too high and not too low). When the value from CFD is out of the acceptable range, the CHTC will be rejected, and the thermal domain will use its own CHTC value based on empirical correlations. This rejection is shown in the graph when the CHTC-value falls down to the base case (no-CFD) value.

The CHTC value of the internal coupling case fluctuates more often compared to the external coupling case. This means that the predictions of CHTC by the internal coupling simulation were rejected more frequently. As shown in Figure 8, the values are high for this type of flow (up to 15 W/m^2K).

The CHTC predictions by external coupling were rejected less frequently (only 3 times in the simplified model and never in the model with frame). During the hottest time the values are around 7 – 12 W/m^2K.

Figure 9 shows CHTC values for the south wall in the model with frame. As the south wall consists of 7 parts (4 glazed sections and 3 frame sections), the values shown in the graph are area-weighted average values.

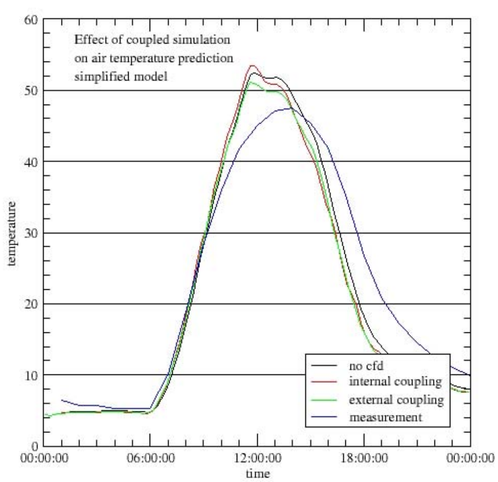


Figure 6. Air temperature comparison (simplified model)

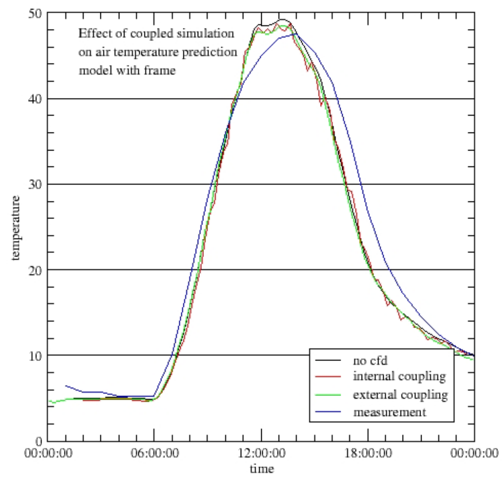


Figure 7. Air temperature comparison (model with frame)

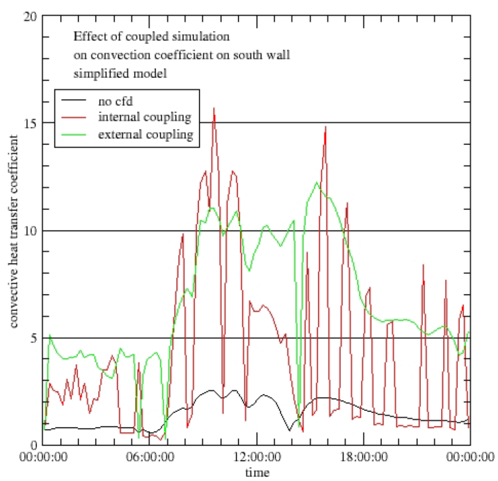


Figure 8. CHTC of south wall (simplified model)

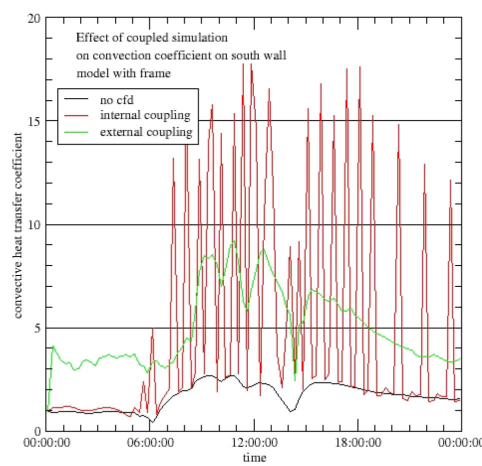


Figure 9. CHTC of south wall (model with frame)

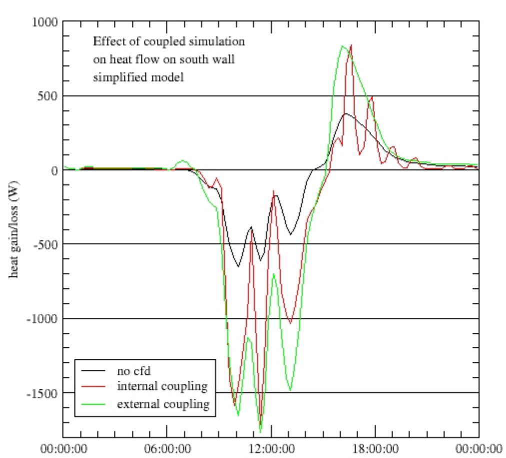


Figure 10. Heat gain/loss of south wall (simplified model).

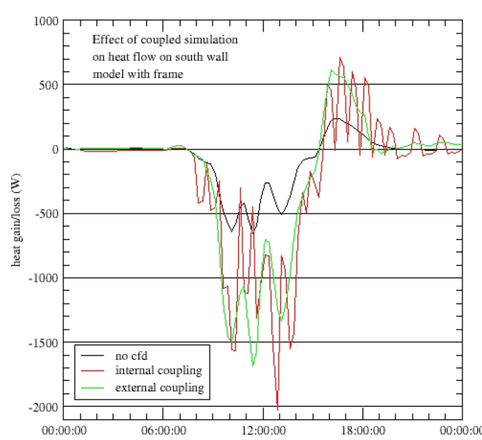


Figure 11. Heat gain/loss of south wall (model with frame).

Most of the time, the internal coupling case prediction of CHTC values are more or less the same as the base case, indicating a rejected CFD predicted value. The prediction of the external coupling case, on the other hand, fluctuates steadily between 6 – 9 W/m^{2}K during the hottest period of the day.

There is no measurement result that can be used to validate the CHTC result. Hiramatsu et. al (1996) however have used Wilke's correlation to estimate the heat flux from the walls. This results in an estimated range of CHTC values between 3.6 – 4.1.

5.2.4 Heat gain/loss

Figures 10 and 11 show the heat gain/loss from the south wall of the atrium. On the model with frame, the values are the sum of the values for the separate parts of the south wall. As expected (based on the CHTC predictions) the coupled simulation enhances the heat flow at the surface. It is consistently higher (either loss or gain) than the base case.

6 Conclusion

The case study shows the potential of the external coupling method. The main advantage of using the external coupling method is the freedom to choose the best software available. With the best software, we can expect more accurate results in a shorter period of time.

The first part of this paper, nevertheless, discusses a procedure to justify the application of such more complex simulation approaches. However, the essence is that, if coupling is judged required, "1+1", in the case of building air flow simulation, can be more than "2".

7 References

Beausoleil-Morrison, I. 2000. The adaptive coupling of heat and air flow modeling within dynamic whole-building simulation, PhD thesis, University of Strathclyde, Glasgow, UK.

Beausoleil-Morrison, I., Clarke, J. A., Denev, J., Macdonald, I.A., Melikov, A., Stankov, P., Further developments in the conflation of CFD and building simulation, Building Simulation 2001, Proceedings of the 7th International IBPSA Conference, Rio de Janeiro, Brazil.

Chen, Q., Peng, X., van Passen, A. H. C. 1995. Prediction of room thermal response by CFD technique with conjugate heat transfer and radiation models, ASHRAE Transactions, Vol. 101, pp. 50 - 60.

Chen, Q., Xu W., 1998. A zero-equation turbulence model for indoor airflow simulation, Energy and Buildings, Vol. 28, No. 2, pp. 137-144

Chen, Q., Glicksman, L. R., Srebric, J. 1999. Simplified Methodology to Factor Room Air Movement and the Impact on Thermal Comfort into Design of Radiative, Convective and Hybrid Heating and Cooling Systems, ASHRAE, American Society of Heating Refrigerating and Air Conditioning, Atlanta, GA, RP-927.

Citherlet, S., Clarke, J. A., Hand, J. 2001. Integration in building physics simulation, Energy and Buildings, Vol. 33, pp. 451 - 461.

Clarke, J. A. 2001. Energy simulation in building design, 2nd edition, Butterworth Heinemann, Oxford and Boston.

Djunaedy, E. 2002. On integration of CFD and building energy simulation: a literature review, Internal Report, FAGO, Faculteit Bouwkunde, Technische Universiteit Eindhoven, Netherlands.

Djunaedy, E., Hensen, J.L.M., Loomans, M.G.L.C. 2003. "Toward External Coupling of Building Energy and Airflow Modeling Programs", ASHRAE Transactions, Vol. 109, No. 2, pp. 771-787.

Djunaedy, E., Hensen, J.L.M., Loomans, M.G.L.C. 2004. "Comparing internal and external run-time coupling of CFD and building energy simulation software ", Proc. Roomvent 2004, Coimbra, Portugal.

Djunaedy, E., Hensen, J.L.M., Loomans, M.G.L.C. 2005. "External Coupling Between CFD And Energy Simulation: Implementation And Validation", ASHRAE Transactions, to be published.

ESRU. 2000. The ESP-r system for building energy simulation, User guide for ESP-r version 9, Energy Systems Research Unit, University of Strathclyde, UK.

Fluent. 2003. Fluent user's guide, Version 6.1, Fluent Inc., NH, USA.

Heiselberg, P., Murakami, S., Roulet, C.-A. 1998. Ventilation of large spaces in buildings, Final Report IEA Annex 26, IEA Energy Conservation in Buildings and Community Systems, Aalborg, Denmark.

Hensen, J. L. M. 1991. On thermal interaction of building structure and heating and ventilating system, PhD thesis, Technische Universiteit Eindhoven, Netherlands.

Hensen, J.L.M., Hamelinck, M.J.H., and Loomans, M.G.L.C. 1996. Modelling approaches for displacement ventilation in offices, Proceedings of the 5th International Conference Roomvent '96, Yokohama.

Hiramatsu, T., Harada, T., Kato, S., Murakami, S., Yoshino, H. 1996. "Study of thermal environment in experimental real-scale atrium", in Roomvent 1996, Proceedings of the 5th International Conference on Airflow in Rooms, Nagoya, Japan, pp. 523-530.

Janak, M., Macdonald, I. 1999. Current state-of-the-art of integrated thermal and lighting simulation and future issues. Building Simulation '99, Proceedings of the 6th International IBPSA Conference, Kyoto, Japan, pp. 1173 - 1180.

Karola, A., Lahtela, H., Hanninen, R., Hitchcock, R., Chen, Q., Dajka, S. Hagstrom, K. 2001. BSPRO COM-Server - interoperability between software tools using industry foundation classes, Building Simulation 2001, Proceedings of the 7th International IBPSA Conference, Rio de Janeiro, Brazil.

Kleijnen, J.P.C., 1997, Sensitivity analysis and related analyses: A review of some statistical techniques, Journal of Statistical Computation and Simulation, Vol. 57, pp. 111 – 142.

Lomas, K. J. 1996. The UK applicability study: an evaluation of thermal simulation programs for passive solar house design, Building and Environment, Vol. 31, pp. 197 - 206.

Lomas, K. J. and Eppel, H., 1992, Sensitivity Analysis Techniques for Building Thermal Simulation Programs, Energy and Buildings, Vol. 19, pp. 21 – 44.

Moser, A., Schalin, A., Off, F., Yuan, X. 1995. Numerical modeling of heat transfer by radiation and convection in an atrium with thermal inertia, ASHRAE Transactions, Vol. 101, pp. 1136 - 1143.

Murakami, S., Yoshino, H., Kato, S., Harada, T., Hiramatsu, T. 1994. Sample data for testing models of air and temperature prediction in large enclosures -- Time dependent data of experimental atrium, Unpublished Report, available online at http://venus.iis.u-tokyo.ac.jp/Annex26/2.pdf. Last visited on 04-May-2004.

Negrao, C.O.R. 1995. Conflation of Computational Fluid Dynamics and Building Thermal Simulation, PhD Thesis, University of Strathclyde, Glasgow, U.K.

Negrao, C. O. R. 1998. "Integration of computational fluid dynamics with building thermal and mass flow simulation", Energy and Buildings, Vol. 27, No. 2, pp. 155-165.

Ozeki, Y., Kato, S., Murakami, S. 1996. "Numerical analysis on flow and temperature fields in experimental real scale atrium", in Roomvent 1996, Proceedings of the 5th International Conference on Airflow in Rooms, Nagoya, Japan, pp. 179-186.

Spitler, J. D., Pedersen, C. O., Fisher, D. E. 1991. Interior convective heat transfer in buildings with large ventilative flow rates, ASHRAE Transactions, Vol. 97, pp. 505 - 515.

Srebric, J., Chen, Q., Glicksman, L. R. 2000. A coupled airflow and energy simulation program for indoor thermal environmental studies, ASHRAE Transactions, Vol. 106, pp. 465 - 476.

Zhai, Z., Chen, Q., Klems, J.H., Haves, P. 2001. Strategies for coupling energy simulations and computational fluid dynamics programs, Building Simulation 2001, Proceedings of the 7th International IBPSA Conference, Rio de Janeiro, Brazil.

Zhai, Z., Chen, Q., Haves, P., Klems, J. H. 2002. "On approaches to couple energy simulation and computational fluid dynamics programs", Building and Environment, Vol. 37, No. 8-9, pp. 857-864.

Zhai, Z. and Chen, Q. 2003. Solution characters of iterative coupling between energy and simulation and CFD programs, accepted for publication in Energy and Buildings.