STEDI – A NEW ASP MODEL FOR WEB-COMPUTING IN THE BUILDING SECTOR

Jordi Pascual *, Guillaume Houzeaux **, Daniel Pérez **, Jordi Cipriano **, Jordi Jiménez ** and Werner Keilholz ***

* Department of building physics, Aiguasol Enginyeria. C/Palau 4, 2-2, Barcelona 08002, Spain + 34 93 342 47 55, + 93 342 47 56, jordi@aiguasol.com

** CIMNE -TERRASSA, Dr. Ullés 2, 3o, Terrassa, 08224, Spain, + 34 93 789 91 69, + 34 93 788 31 10, houzeaux@cimne.upc.es

*** CSTB –Software Division, BP 209, 06904, Sophia-Antipolis, France, + 33 493956700, + 33 493956733, werner@cstb.fr

Key Words – Web computing; Thermal analysis; Transient simulations; CFD simulations.

Abstract. The use of internet based tools as a support to traditional methods for design, analysis and surveillance in the building sector will reduce costs, while increasing at the same time the market value of the end product. STEDI ASP model aims to reach these benefits helping engineers and architects to design more energy efficient buildings. This project adapts two simulation codes, a dynamic transient calculator (TRNSYS) and a computational fluid dynamics (CFD) code (FAVENT), with a friendly user interface in a web-computing platform (GiD). The service is based on an ASP model which provides all the advantages of these services: the easy, low cost and safe access to complex thermal simulation, without the need for purchasing expensive software or hardware nor for maintaining a technical staff. With the STEDI model it will be possible to predict the annual building thermal behaviour, to calculate the monthly energy demand and, finally, to simulate the airflow and the influence of the shape during some representative days.

1. INTRODUCTION

The construction sector is one of the main energy spenders sectors in Europe, 385 Mtoe in 2000 (source domestic & tertiary Eurostat). The importance of e-work in the AEC sector is growing exponentially. Nevertheless it is a fact that web computing is still at its infancy. The need for providing this sector with modern IT tools which allows implementing bioclimatic concepts is an urgent priority. This is a challenging goal, as most engineers and architects are not familiar with modern IT techniques hardly adapt to the rapid changes in technology.

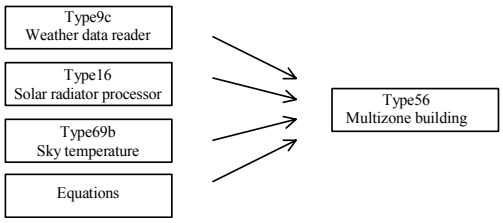
As it will be shown, STEDI service couples a thermal and a CFD simulation codes introducing them to the user with a single and friendly interface. Two powerful codes have been used: the dynamic transient calculator TRNSYS and the CFD code FAVENT. The model is implemented as an ASP service with all the advantages of such a kind of services: the easy, low cost and safe access to sophisticated thermal simulation applications, without purchasing expensive software nor maintaining a technical staff.

Considering STEDI from a point of view of an easy to use tool to check the buildings on the initial phases of the design, some new features of the pre and post processor GiD were implemented. The new developed interface allows the user to easily define in a graphical way all the needed information by importing CAD file. Moreover a wizard methodology was developed to simplify all the required steps so that it can be used by non expertise professionals.

2. TRNSYS MODEL

2.1 Reference Model

STEDI has been developed as a pre checking tool for the first phases of the building design process. That means, the needed information required by the multizone building is coming from the standard subroutines used in TRNSYS and some equations. The basic TRNSYS system implemented within STEDI uses the multizone model (Type56) with the most common subroutines used for thermal building simulations as it is shown in the next figure:



In order to simplify the process of introducing the simulation data, not all the existing possibilities within Type 56 model are allowed. In most of the cases, the user have to choose between some predefined constant data or schedules.

Nevertheless there are some parameters which are susceptible to be freely selected by the user in order to have coherent results with their design simulations. The equations allows the user to previously define some inputs which will be introduced through the graphical interface, like e.g. the start and stop times for the heating and cooling systems. In a similar way, the user has the possibility to choose the global settings of the simulation.

2.2 Implemented new subroutines

Based on the foreseen TRNSYS model, some new features have been implemented. In order to obtain the proper outputs and to allow the use of these transient results as inputs for the CFD simulation, two new subroutines were developed.

On the one hand the so-called Type 919 subroutine was created to offer the results the end user had previously defined. The use of this new type allows to know the daily averages values of the external air temperature, and of the heating and cooling loads per square meter for any zone of the building. Furthermore the global heating and cooling loads for the entire building is also offered among other characteristic results.

On the other hand the new Type 920 subroutine was created to allow the end users to couple TRNSYS with the CFD code. One of the main advantages of STEDI tool, with regard to other existing softwares, is the possibility to run a CFD simulation with predefined boundary conditions. The required boundary conditions for using the CFD code are not necessarily estimated data or data coming from a monitoring process but there also exists the possibility to import the TRNSYS outputs. This allows non-experienced users to easily implement the CFD simulation by using the previously calculated conditions. To do so, the users must choose the significant instants of time which are used to save a file with the indoor walls temperatures.

3. CFD MODEL

3.1 Physical problem

We present in this section the governing equations considered to model internal as well as external ventilation problems. The fluid is air and is assumed to be slightly compressible. This implies that the momentum and mass conservation equations (which involve the velocity **u** and pressure p) are coupled to the heat equation (which involves the temperature T). The approximation which is commonly used to couple these three variables is the Boussinesq approximation, and is valid for temperature differences of the order of few degrees. It consists in expanding the density ρ in Taylor series around a reference density ρ_0 measured at a reference temperature T_0 such that:

$$\rho \approx \rho_0[1-\beta(T-T_0)],$$

where β is the thermal expansion coefficient.

The state of the air flows under consideration is generally turbulent, the Reynolds number being of the order of 10^5 up to 10^7, so that turbulence modelling is necessary. In the perspective of the solution of large scale problems, ensemble averaging (also called Reynolds averaging) is performed to filter the Navier-

Stokes equations, that is to decompose the flow variables into average and fluctuation components, the latter representing the turbulence desviation around the average. This decomposition introduces a new term in the momentum equations called the Reynolds stress tensor involving the correlations between the fluctuation components, as well as an additional term in the heat equation. The Reynolds stress tensor is modeled by the Boussinesq approximation, while a "similar" approximation is used for the heat equation. The resulting system is called the RANS equations and reads:

$$\rho_0 \frac{\partial \mathbf{u}}{\partial t} + \rho_0 \left(\mathbf{u}\cdot\nabla\right)\mathbf{u} - 2\nabla\cdot\left[(\mu+\mu_t)\,\varepsilon(\mathbf{u})\right] + \nabla p = \rho_0 \mathbf{f}$$

$$\nabla\cdot\mathbf{u} = 0$$

$$\rho_0 c_p \left[\frac{\partial T}{\partial t} + \mathbf{u}\cdot\nabla T\right] - \nabla\cdot\left[(k+k_t)\nabla T\right] = 0$$

In the equations above, the physical properties are the molecular viscosity μ, the specific heat c_p and the thermal conductivity k. Two numerical parameters have been introduced by the turbulence modeling, namely the turbulent viscosity μ_t which is a new unknown of the problem, and the turbulent thermal conductivity k_t which is given as a function of μ_t

$$k_t = \frac{c_p \mu_t}{Pr_t},$$

where Pr_t is an dimensionless parameter called the turbulent Prandtl number (which varies 0.9÷1.0 for air). ε is the velocity symmetrical gradient tensor

$$\varepsilon(\mathbf{u}) = \frac{1}{2}\left(\nabla\mathbf{u} + \nabla\mathbf{u}^T\right),$$

and **f** is the vector of body forces per unit of mass.
It contains gravity acceleration and the buoyancy terms coming from the boussinesq assumption, that is:

$$\mathbf{f} = \mathbf{g} - \mathbf{g}\beta(T - T_0),$$

where **g** is the gravity acceleration.

3.2 Turbulence modeling

By performing the ensemble averaging on the Navier-Stokes and heat equations, and applying the Boussinesq approximation for the Reynolds stress, we have introduced a new unknown, the turbulent viscosity, that needs to be modeled.

While the two-equation k-ε turbulence model is extensively used in the simulation of ventilation problems (Chen, 1995) (Xu and Chen, 2001), we chose a more simple one-equation turbulence model, namely the Spalart-Allmaras (SA) model (Spalart and Allmaras, 1992). It consists of a single partial differential equation which solves directly for the eddy viscosity:

$$\frac{\partial \nu_t}{\partial t} + \mathbf{u}\cdot\nabla\nu_t + c_{w1} f_w \frac{\nu_t^2}{d^2} - \frac{1}{\sigma}\nabla\cdot\left[(\nu+\nu_t)\nabla\nu_t\right] = \frac{c_{b2}}{\sigma}(\nabla\nu_t)^2 + c_{b1} S \nu_t$$

where c_{w1}, σ, c_{b1} and c_{b2} are constants of the model. S is the module of the vorticity, d is the distance to the wall and f_w is a function of the vorticity, the distance to the wall, and the eddy viscosity itself. The version displayed here is to be used for high-Reynolds number flows. A low-Reynolds number version, which includes additional terms, is also available. It is not shown here for the sake of clarity.

3.3 Boundary conditions

In order to close the complete system of equations (RANS+heat+SA), initial and boundary conditions must be provided. The boundary conditions used in this work are of Dirichlet type (for inflows) and Neumann type (for outflows). The conditions used on the walls depend on the physics of the problem and are explained next.

For low Reynolds number flows, the governing equations are integrated up to the wall. For high Reynolds number flows, the wall function approach is used for the velocity, temperature and eddy viscosity (Houzeaux and Codina, 2003).

3.4 Numerical model

The equations are solved using a stabilized finite element method. The stabilization technique used in this code is the Algebraic Subgrid Scale model (ASGS) of (Codina, 1998) and originally proposed in (Hugues, 1995).

3.5 Coupling Dynamic model and CFD

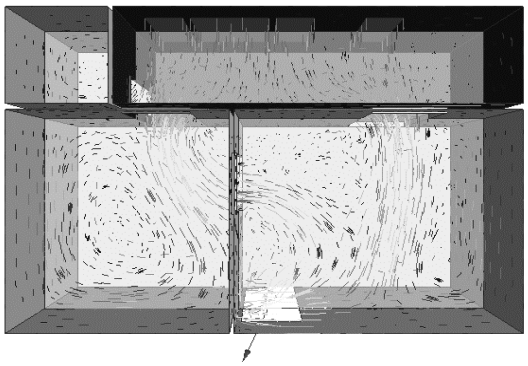
The coupling between dynamic and CFD models can be understood as a domain decomposition method. On the one hand, temperature fluxes are prescribed as inputs of the dynamic model, the output being the temperatures on the walls of the building.

These temperatures are then used as boundary conditions for the CFD model. The temperature fluxes could then be calculated CFD simulation results and taken as new inputs for the dynamic model. This sets up an iterative process between the two simulation models that should be carried out until convergence. In our case, we just assume that the temperatures calculated by the dynamic model do not change within the time range of the CFD simulation, so that they can be considered constant.

3.6 A Numerical example

We present the results of the simulation of a forced convection problem. A flat is ventilated with a velocity of 1 m/s, as shown in the following figure. The corresponding Reynolds number is $Re=1.3 \times 10^5$, the characteristic length being that of the biggest window. In order to solve this problem within a reasonable time, the law of the wall is applied, assuming that the computational wall is located at a distance of 4 cm from the real wall.

Next figure shows the velocity vectors and enable to appreciate the recirculation obtained in the rooms.



4. GRAPHICAL PRE AND POST PROCESSOR

4.1 Main features

GiD is an interactive graphical user interface used for the definition, preparation and visualization of all the data related to a numerical simulation. This data includes the definition of the geometry, materials, conditions, solution information and other parameters. The program can also generate a mesh for finite element, finite volume or finite difference analysis and write the information for a numerical simulation program in its desired format.

GiD works, when defining the geometry, similar to a CAD system but with some differences. The most important one is that the geometry is constructed in a hierarchical mode. All materials, conditions and other parameters can also be defined on the geometry without having any knowledge of the mesh: the mesh is done once the problem has been fully defined.

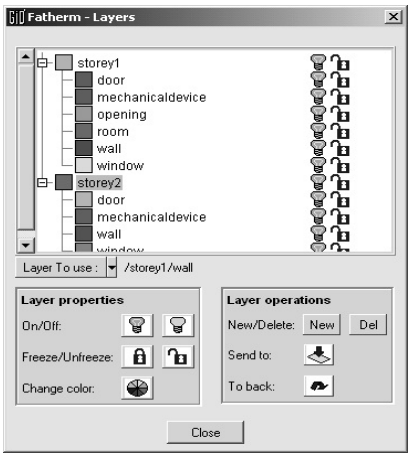
Full graphic visualization of the geometry, mesh and conditions is available for comprehensive checking of the model. More comprehensive graphic visualization features are provided to evaluate the solution results after the analysis run.

Another necessary important features of GiD for the STEDI project is its capability to re-use the geometry information from existing CAD drawings. This enables the user to rapidly convert existing projects to a format ready for simulation, which saves an incredible amount of project time. A ProcServer, which is totally integrated within GiD, allows to the end-users the calculation with a remote server.

4.2 Customisation to the needs of end-users

Taking into account all the end-users requirements and taking advantage of the customisation capacity of GiD, a customisation to adapt it to TRNSYS software and CFD model was carried out.

One of the first modifications was the creation of a layer manager specially adapted to buildings. This new layer manager allows the hierarchic organization of the different building elements. See next figure:



In the case of the TRNSYS software, particular emphasis was put on the development of user friendly windows for the building data input. The first package of algorithms is dedicated to the reading of 2D floors, imported from DXF files, and the last package was focused on the generation of the data files to run the

simulation. All these algorithms has been included into a wizard, which guides the end-user step by step, avoiding errors and facilitating the use of the software.

The second part of the development carried out in this project was focused on the CFD model. The aims of this development was focused in the same direction than in the case of the TRNSYS software, to obtain a user-friendly software which allows end-users to simulate the thermal behaviour of their building using a CFD calculation. The inputs for this calculation can be directly imported from the outputs of the dynamic transient simulation. And the geometry is the same than was generated to calculate with TRNSYS model. All the other need data to define the problem that must be solve by the CFD model has been decreased. The data to be introduced by the users are very few and understandable for non-specialized in CFD codes.

The communication among these two STEDI components follows the ASP structure, which means the graphical browser creates the geometry, assigns the conditions and the problem data and then it sends these inputs, through internet, to the remote server. Within this server both solvers are installed, then, the simulation is run and outputs are automatically returned to GiD and the user can see these outputs by means of the graphical possibilities of GiD.

5. CONCLUSIONS

The use of STEDI will help end-users to design buildings with good thermal behavior. Furthermore, the recent or future implementations of thermal regulations within this market all over the European countries will force the use of such simulations.

The STEDI ASP service will help to overcome these fences through all the implicit advantages of these kind of systems. That means, end users will have the possibility to use two powerful codes of thermal behavior on buildings.

The STEDI model has major advantages compared to other thermal building simulation methodologies. By using the STEDI service it is possible to predict the annual thermal evolution of each building zone, to calculate the monthly energy demand and, finally, to simulate the airflow during some representative days. This is a crucial fact for the success of STEDI because of the implicit possibilities of it. The standard available thermal simulation software is only capable to carry out one of these analyses, but they can't combine them simultaneously in only one software package. That means to offer a complete solution for architects and constructors (end users of AEC sector) but also for thermal consultants who will be able to have more reliable results, even considering the simplicity of the tool, because of the possibility to use simulated outputs as inputs for the CFD calculations and to perform iteration processes.

REFERENCES

Codina R. (1998). A comparison of some finite element methods for solving the diffusion-convection-reaction equation, *Computer Methods in Applied Mechanics and Engineering*. 156, 185-210.

Codina R. and Soto O. (1999). Finite element implementation of two-equation and algebraic stress turbulence models for steady incompressible flows *Int. J. Num. Meth. Fluids.* 30, 309-334.

Chen Q. (1995). Comparison of different k-ε models for indoor airflow computations*, Numerical Heat Transfer, Part B: Fundamentals*. 28, 353-369.

Duffie and Beckman (1991). Solar Engineering of thermal processes.

Houzeaux G. and Codina R. (2003). A Dirichlet / Neumann domain decomposition method for incompressible turbulent flows on overlapping subdomains, *Comput. Fluids*, In press.

Hughes T. J. R. (1995). Multiscale phenomena: Green's function, the Dirichlet to Neumann formulation, subgrid scale models, bubbles and the origin of stabilized formulations, *Computer Methods in Applied Mechanics and Engineering*. 127, 387-401.

SIMCAD for TRNSYS, The CAD Tools for Building Simulation, CSTB, (2000)

Spalart P.R. and Allmaras S.R (1992). A One-equation Turbulence Model for Aerodynamic Flows. *AIAA Paper 92-0439*.

TRNSYS, Trnsys System Simulations, Version 15,

Xu W. and Chen Q. (2001) A two-layer turbulence model for simulating indoor airflow - Part I. Model development, *Energy and Buildings*, 33 (6), 613