TransAT Report Series
– Presentations –

TransAT for Environmental Engineering
Computational Wind Engineering Problems

Ascomp Switzerland
Edited by: Dr Mathieu Labois
Release date: Sep, 2014.
References: TRS-P / 05-2014
# Table of Contents

1. The Numerical Approach in Transat ......................................................... 2
2. Sudden Rotation of Cranes Under Flow Gusts ........................................ 2
3. LES of Turbulent Flow Past a Cylindrical tower ................................... 5
4. RANS & LES of Turbulent Flow Past a Bridge ....................................... 6
5. Eindhoven Building ............................................................................... 8
   5.1 Description ...................................................................................... 8
   5.2 Results and Discussion ...................................................................... 9
6. 3D Flow Around the Must Matrix of Buildings ...................................... 10
   6.1 Description ...................................................................................... 10
   6.2 Results and Discussion ...................................................................... 13
7. TEXAS AM TTU Building ........................................................................ 13
   7.1 Description ...................................................................................... 13
   7.2 Results and Discussion ...................................................................... 14
8. Flow & Pollution Around an Incinerator ................................................. 16
   8.1 Description ...................................................................................... 16
   8.2 Simulation Results ........................................................................... 18
9. Conclusions ........................................................................................... 18
Abstract:
ASCOMP provides consulting services in wind and environmental engineering, including urban pollution dispersion from cars, industry or even accidental gas releases. Our studies are here based on detailed 3D simulations using TransAT HPC. The use of this version of our CFD platform is motivated by two important elements: (i) rapid meshing of industrial wind engineering and pollution problems using the immersed surface technique with block mesh refinement, and (ii) advanced turbulence modelling based on URANS, VLES or LES. CPU intensive problems could be treated using VLES instead of LES. Selected examples are described below, without details though.

1. The Numerical Approach in Transat
The CMFD code TransAT© deals with complex-fluids, single and multiphase flows of industrial relevance. It adopts an original meshing technology among other existing commercial codes, known as the Immersed Surfaces Technology (IST). Mesh generation can also be achieved either using traditional Boundary Fitted Coordinates (BFC), with help of external meshing softwares like GRIDGEN, ICEM. The IST Technology combined with Block-Mesh Refinement (BMR) capability offers great advantages for complex geometries modelling. This code is particularly suited for complex fluid flows and offers powerful solution algorithms suited for parallel processing -using both MPI and OpenMP protocols, a wide portfolio of turbulence models and approaches -including LES and VLES, combustion and reactive flows, multiphase physics with conjugate heat and mass transfer.

The three segments in which the ASCOMP’s code excels are: (1) advanced RANS turbulence models like Explicit Algebraic Stress Models, EASM, (2) Multiphase flow heat transfer, and (3) Scale Resolving Turbulence strategies like LES and its sub-variants including V-LES short for Very Large-Eddy Simulation, (Labois and Lakehal, 2011).

2. Sudden Rotation of Cranes Under Flow Gusts

In this challenging example, the focus is on estimating wind forces that may lead to an accidental rotation of a crane placed in the yard of a building site under the action of wind gusts. Such studies now are mandatory for all in-situ urban constructions under EU laws since a couple of years now. Classical approaches to the problem (based on 3D calculations) rely on the re-construction of the wind energy spectra from steady state RANS simulations. This has shown to be rather limited in most cases, suggesting the use of real unsteady simulations, either using the U-RANS approach or the LES. For high Re numbers, a well-resolved LES requires very large grids and small time steps. One way to avoid these constraints in practical applications (when a full wind rose is required) is to resort to less expensive techniques such as the V-LES technique presented in this paper.
The problem chosen here is related to a real in-situ construction yard located in the city centre of Mulhouse (France). The crane fleche is shown in blue in Fig. 2; the height of each building is 24.5m and 33.5m for the two parallelepiped buildings, and 72.5m for the round one. In the example treated here (Fig. 2), the experimental investigations carried out CSTB Nantes (France) indicate that the largest energetic scales correspond to about half-length of the crane. This allows fixing or limiting the under-resolved scales by reference to this detail. The flow was then simulated with unsteady turbulent conditions for six wind directions, using the physical model described previously. The grid is of IST type, refined using Block Mesh Refinements (3 blocks) to better resolve the boundary layers. The shedding motions obtained by the V-LES are shown in Fig. 3.

The figure clearly shows that the model predicts, as was to be expected, flow scales of dimensions equivalent to the building sizes; smaller flow detailed are filtered and modelled. It is perhaps interesting to note that the flow has also been simulated using U-RANS approach, which provided indeed various shedding scales, but less than with the V-LES. The PSD spectra shown in Fig. 6 obtained with the V-LES (over 2 minutes of real time: with $\Delta t = 0.05s$) depicts clearly the dominant modes that can cause the accidental turning of the crane, at that specific wind direction (145 deg.). The PSD spectrum shows that the model predicts scales down to 0.6Hz.
Figure 4: Energy spectra for three velocity components returned by V-LES.

Figure 5: Energy spectra for three velocity components returned by V-LES.
3. LES of Turbulent Flow Past a Cylindrical Tower

Figure 6: Turbulent structures in the wake of the cylinder (isovalues of $u'v'$).

Figure 7: Drag force PSD signals from the LES simulation.

In this example, we have simulated the turbulent flow past a cylindrical tower at $Re=2.105$. Wind tunnel experiments were performed by Barré and Barnaud [7]. A cylindrical finite tower of 14 cm diameter was placed in an atmospheric wind tunnel with oncoming air flowing at a bulk velocity of 20 km/h and a turbulence intensity of $I = 2\%$. The flow pattern is obviously unsteady, and use was therefore made of LES. The sub-grid scale (SGS) model of Nicoud and Ducros [8] was employed, together with with Schuman's wall functions. The domain is 8x4x1.3m, with the smallest cells being 1.75x5mm. The IST/BMR grid contains 2.7 million cells, with 3 levels of BMR clustered around the boundary layer to capture small-scale motions. The mesh was created in 20 minutes. Two days of simulation on a quad-core PC were necessary to predict a statistically convergent data basis.

LES provides indeed a detailed flow solution as shown in Fig. 6. The power density spectrum for drag force as obtained by LES is shown in Fig. 7, where the shedding frequency is revealed at 0.07 Hz. To validate our simulation, mean pressure coefficients and root mean square (RMS) pressure coefficients were compared to experimental data from Barré and Barnaud [6]. As can be judged from Fig. 8, both the mean and RMS Cp at cylinder mid-height are in excellent agreement with the measurement data.
4. RANS & LES of Turbulent Flow Past a Bridge

The viaduct of Millau (Fig. 9) is part of the A75 motorway linking Paris and Barcelona. The proposed design is a multi-span cable-stayed bridge crossing the two kilometer wide valley at a maximum deck height of approximately 300m and a maximum pylon height of approximately 340m. The great height of the structure also leads to severe wind loading and thus presents a challenging aerodynamic design task. Extensive tests on an early design of the Millau bridge deck were performed at the CSTB of Nantes. The results of the tests are used herein for comparison with the RANS and LES calculations of the flow.

Time averaged flow patterns obtained from 2D RANS simulations depicted in Fig. 12 show the dependence of the flow separation on the angle of attack. In the negative angle-of-attack group, the flow separates only at the downstream lower corner. At zero and positive angle of attacks, the recirculation occurs at the upstream upper corner and extends up to 40% of the deck for $\alpha = 6\text{deg}$. On the windward bottom face, the flow reattaches for all $\alpha$, while the reality shows that it separates intermittently (see LES and data). This result was to be expected from steady RANS simulations using the standard $k-\varepsilon$ model combined with wall functions.
Figure 9: Millau bridge, South west of France

Figure 10 compares the average mean force coefficients (moment, drag and lift), normalized by model dimensions \( B = 388\text{mm} \) and \( D = 60.2\text{mm} \), for \(-6 < \alpha < 6\). Results are compared to wind tunnel data of CSTB Nantes, France. The agreement is good for all the mean forces. The reduction of the mean moment force at 3 & 6 deg. angles is not well predicted because the flow separation under these conditions is intermittent and can only be predicted by means of LES (see next section). LES were performed in 3D using the SGS model of Nicoud and Ducros [8] combined with wall functions. The IST/BMR grid contains 2.1 million cells, with 2 levels of BMR. The LES is capable to provide a detailed flow picture as displayed in Fig. 11, with vortex shedding patterns faithfully comparable to the data (soap film technique is due to Morgenthal [10]).

![Graph](image)

Figure 10: Mean drag lift & moment coefficients for various \( \alpha \).

The normalized force coefficients for \( \alpha = 0 \) are shown on Fig. 10 as a function of non-dimensional time \( t^+ \). While \( C_L \) is slightly over-predicted, \( C_D \) compare very well to the data, way better than Morgenthal’s Vortex Methods results.
5. Eindhoven Building

5.1 Description

Figure 11: Vortical structures at various angles of attack: 3D LES vs. visualizations of Morgenthal [10].

Figure 12: Picture of the main building of Eindhoven University of Technology.
This test case is interesting since it directly deals with unsteady flows past a real-scale building, for different wind directions. Measurements were performed for two wind incidence angles: 225 and 270 Degr. The building shown in Figure 12 is the main one at the technology university campus.

The simulations were conducted using TransAT for the two wind incidence angles, using V-LES, with a filter width equivalent to 10th of the building height. LES for this case is out of reach of computing resources. The most important issue here is to generate the inflow turbulence properly; this was achieved using the TransAT wind generator module, requiring specification of the correlation length scales. The test case was thus taken as a ‘feasibility exercise’ for V-LES. The objective was to determine the unsteady forces on the structure for different wind incidences, and analyse the power spectra of the wind.

5.2 Results and Discussion

Instants of simulation results for the 270 Deg. incidence are shown in Fig. 13 below. The image shows the variation of pressure loads on the building façade, proving that unsteady wind structures are predicted according to wind inflow characteristics. Note too that the recirculation in the 270 deg. was found to very large compared to the lateral one, 225 Deg.

The pressure fluctuations on one point of the building façade with time as predicted with V-LES are plotted in Fig. 14. The plot shows a very high variation of pressure on that point, indicating a very high RMS of pressure coefficient, which cannot be inferred from steady-state simulations, or even simple U-RANS.
What is very interesting from these V-LES simulations is the power spectra for drag and lateral forces on the buildings; these are plotted in Fig. 15 for both wind incidences. Interestingly, the results indicate that the mean lateral lift in the 270 deg. is zero, but the instantaneous one is very large; larger than the drag. The mean lateral lift in the 225 deg. is not zero, and is almost equal to the mean drag.

6. 3D Flow around the Must Matrix of Buildings

6.1 Description

The MUST in-situ experiment depicted Fig. 16 was conducted with the objective to provide a full-scale high-Re (up to Re = 500,000) wind-flow database for model validation and testing (Yee and Biltoft, 2004). The experiment was designed to document the effects of an array of large building-like obstacles on flow and dispersion over a range of atmospheric stability regimes. MUST was conducted at the DPG Horizontal Grid located in the Great Basin desert of western Utah, USA. The test site is predominately flat, with sparse grease-wood and sagebrush of 0.5 to 1.0 m height as the predominant vegetation. The roughness length z0 outside the MUST array is within 2 to 4 cm. The MUST urban roughness layer was created using containers (each 12.2 m long, 2.42 m wide, and 2.54 m high) positioned around the center of Horizontal Grid. As shown in Fig.16, the conex containers were arranged in a 12 by 10 array.
This test case has been selected by the COST 732 Community for model validation. It has been selected by ASCOMP as an additional test case to validate the RANS version of the code TransAT, using the Immersed Surface Technology.

Briefly, the flow was simulated in steady state with the K-ε model with Wall Functions accounting for surface roughness. The grid consists of $261 \times 177 \times 25$ Cartesian cells. The IST technique has been employed to map the obstacles into the Cartesian mesh. The domain
extension for the 0 Deg. incidence was set as follows: −144m < X < 170m; −150m < Y < 150m; and Zmax = 21m, and −150m < Xmax < 150m; −150m < Y max < 150m; and Zmax = 21m for the 45 Deg. incidence. The results presented in this Section refer to the 0 Deg. incidence, only, and are qualitatively similar to those obtained recently within the scope of the COST 732 project (Di Sabatino and Buccolieri, 2007).

Figure 18: U-velocity and TKE vertical profiles for 0 Deg. Flow incidence.
6.2 Results and Discussion

Figure 18 presents the flow streamlines and U-velocity iso-contours obtained for both wind direction (0 and 45 Degs.). The off-set position of some of the buildings is seen to affect the wake flow. Wakes are seen to develop past all obstacles (negative U-velocity iso-contours). Figure 18 compare selected time-averaged U-velocity and TKE profiles with the data, for the 0-Deg. flow incidence (the data are taken at the points depicted in Fig. 25). The results are within the predictive-performance of all other reported simulations (Di Sabatino and Buccolieri, 2007), using the standard K-E model with wall func- tions. Surprisingly, the backflow (recirculation) shown in the three-upper right panels is well predicted by the model. Somewhat the boundary layer profiles seem to present a deficit in the core flow as compared to the data, which can be explained by the coarse resolution of the flow field for this very high Re. The TKE profiles are undepredicted by the model, which is not surprising. The same is true for the 45-Deg. flow incidence (results now shown), where the angle formed between the flow-streamlines and the grid increases the rate of numerical diffusion.

7. TEXAS AM TTU Building

7.1 Description

The TTU building is a 4 m high, 13.7 m wide and 9.1 m long rectangular box. The simulation domain size was set to be larger than that used by Mochida et al. (1993). The height of the computational domain was set to be 40 m. The upstream distance of the inflow from the building was set to be 5.6H, where H is the height of the building, and 12H downstream of the building. The near-wall grid size was also set to match the values used by Mochida et al. (1993) closely (0.08m). Six different cases were run to clarify the following aspects: The effect of Reynolds number, of inflow turbulence intensity, of the choice of time step and of the filter width for VLES simulation. By default a value of 1 m was chosen for all cases. However, two cases with filter width = 0.3 m and very large filter width were also performed for comparison.

The filter–based inflow generator (Klein et al., 2003) in TransAT has been used to provide unsteady inflow conditions. The following correlation lengths were specified as inputs; streamwise 10 m, vertical and spanwise 3 m. The non-uniform turbulence profiles feature was not used in these simulations since the implementation needs to be further improved in order to take into account the drastically bunched grids at the inflow. The new method implemented, however, is based on direct numerical simulation (DNS) database (Iwamoto et al., 2002) and the user need not specify any

<table>
<thead>
<tr>
<th>N</th>
<th>Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Base</td>
<td>VLES with real Reynolds number</td>
</tr>
<tr>
<td>2</td>
<td>RE100</td>
<td>Viscosity increased by factor of 100</td>
</tr>
<tr>
<td>3</td>
<td>RE1000</td>
<td>Viscosity increased by factor of 1000</td>
</tr>
<tr>
<td>4</td>
<td>RE100DT</td>
<td>RE100 with lower time step</td>
</tr>
<tr>
<td>5</td>
<td>RE100MI</td>
<td>RE100 with medium turbulence intensity</td>
</tr>
<tr>
<td>6</td>
<td>RE100HI</td>
<td>RE100 with high turbulence intensity (30%)</td>
</tr>
<tr>
<td>7</td>
<td>RE100HIFG</td>
<td>RE100HI with 2 times more grid points</td>
</tr>
<tr>
<td>8</td>
<td>RE100HIFG(VLES .3m)</td>
<td>RE100HIFG with smaller filter width</td>
</tr>
<tr>
<td>8</td>
<td>RE100HIFG (URANS)</td>
<td>RE100HIFG with very large filter width</td>
</tr>
</tbody>
</table>

Table 1: Test Cases studied
correlation lengths or turbulence intensities. The DNS database is for $Re_\tau = 642$, and $Re_\delta = 24272$. As DNS data for higher Reynolds numbers are available this database will be updated. Simulation results show that this is not the essential feature that is needed to predict pressure fluctuations on building accurately.

Figure 19: Turbulence profiles for low and high intensity simulations

Figure 19 shows the inflow turbulence intensity profiles for the high intensity cases (described in Table 1 above). Low intensity has a target intensity of 13% turbulence, the Medium intensity case has a target turbulence intensity of 20% and the high intensity has a target intensity of 30%. However, a posteriori, based on TransAT unsteady generator, the final intensities were verified to be lower; 8%, 15%, and 21% for the low, medium, and high intensity cases, respectively. The difference is primarily because of ensuring that the inflow perturbations sum up to zero over the whole inflow field. Since the pressure fluctuation amplitudes depend directly on the velocity fluctuations, this has to be kept in mind for comparison.

7.2 Results and Discussion

Figure 20 shows the mean pressure contours for the three cases with different Reynolds numbers, namely Base and RE100 cases. No significant differences are visible in the mean pressure contours. The mean pressure coefficient was calculated using the mean pressure. Note the outflow pressure which was set to zero is used as the reference pressure to calculate $C_{p,mean}$. The averaging from performed from $\Delta t = 25 - 750$ s.

Figure 20: Mean velocity (upper panels) & pressure contours (lower panels). Left: Base; right: RE100DT.
Comparison of $C_p$mean with experimental data and simulations of Mochida et al. (1993) and Nozawa & Tamura (2002) is presented in Fig. 21. The reference data has been extracted from the graphs presented in their respective papers. It can be seen that the results of Mochida et al. (1993) match the experimental data the best. Especially noteworthy is the rate of pressure recovery after the recirculation at the corner. In the experiments and the results of Mochida et al. (1993) an almost complete recovery is achieved before the aft corner is reached. TransAT results are reasonable although full recovery is not achieved (the recirculation zone size is over predicted). This results in a small readjustment in the mean pressure just before the aft corner. The results of Nozawa & Tamura (2002) do not show any pressure recovery along the horizontal surface implying a very large recirculation zone (see also Figure 31(a) in their paper). Note that only their smooth wall case is considered. In their rough wall case (Figure 31(c)) the recirculation zone is much shorter and therefore the pressure recovery matches the experimental data better (see Figure 33(a)).

The $C_p$rms comparison is presented in Fig. 22, where panel (a) just shows the experimental data collected from both the papers. The two measurements of Tamura et al. (1998) at high turbulence intensities stand out from the other measurements. Overall significant scatter in the data exists. The high intensity measurements are not considered for comparison. Figure 22(b) shows the comparison between the experimental and simulation data. The results of Nozawa & Tamura (2002) are far away from the experimental data. Even their results for the higher roughness factors (see Figure 22(b)) are not close to experimental data. For the high roughness case, $C_p$rms is significantly lower on the horizontal wall (a closer look at the mean pressure recovery in Figure 22(a) shows that the recovery is incomplete).

The results of Mochida et al. (1993) are quite better in the horizontal surface and also overall. This is primarily due to a good reproduction of the recirculation and pressure recovery. However, significant differences exist on the stagnation surface. It is possible that the RMS pressure coefficient is less dependent on the inflow turbulence intensity in the horizontal surface as compared to the front of the building. The experimental data show a lot of differences in the front part of the building (A-B section), whereas they show similar behaviour in the horizontal surface.

TransAT results lie somewhat in between the results of Nozawa & Tamura (2002) and Mochida et al. (1993). The RMS coefficient at the A-B section depends strongly on the incoming turbulence intensity. It seems that if the low pressure zone (recirculation zone) is much flatter then the wall can feel higher pressure fluctuations from the free stream flow. Note, in particular, that the RE100HI results of TransAT are quite similar to that of Nozawa & Tamura (2002) (see Figure 22(b)). Overall the behaviour of $C_p$ms is less accurate in the horizontal surface. Further simulations are required to verify this hypothesis, such that the flow near the front corner and the adjoining horizontal surface is much better resolved (also in the sense of simulating a reduced Reynolds number).
A rise in Cprms just before the aft corner is observed in TransAT results as well as in the results of Nozawa & Tamura (2002). This is primarily because of the incomplete recovery of the mean pressure so that an abrupt adjustment is needed for the mean pressure just before the aft corner, as discussed before.

It is clear that the recirculation zone near the front corner and the later mean pressure recovery needs to be captured better in order to obtain better results for the RMS pressure coefficient. To obtain this a more refined grid is required along with simulations at lower Reynolds numbers. A turbulence intensity of at least 20% is required (in the case of TransAT an effective intensity). Fluctuations based on a turbulent boundary layer profile should automatically have the correct intensity.

In conclusion, the unsteady pressure loads on buildings and other structures can indeed be captured using an approach like the VLES method.

8. Flow & Pollution Around an Incinerator

8.1 Description
The context of this simulation is new, in that it has as an objective to evaluate over time the dispersion of a pollutant emanating from an industrial incinerator in the city centre (Fig. 23). The incinerator is located three blocks east of the Technopark towards the City Centre.
The specific goal is to analyze the pollution dispersion in the vicinity of a building under design nearby.

![Image](image1.png)

**Figure 23:** The city incinerator in Zurich West seen from neighboring building

![Image](image2.png)

**Figure 24:** Pollution dispersion around the Technopark (the buildings are drawn with Google-Earth).

For the purpose, we have developed a pseudo-transient approach consisting in solving the flow within an interval of time (up to one year in the past), using local meteorological data. In this aim, we have used past meteorological data for two days (from 31 Dec. 2002 at 6am to 02 Jan. 2003 at 3pm), including wind speed at 10m height, thermal stratification, wall shear, heat flux, etc. Inflow boundary conditions are then variable, according to these data; other boundaries conditions are velocity outflow conditions. The flow was simulated using the k-epsilon model with wall functions for wind conditions taken at every hour, in steady state, for a grid consisting of 129 x 129 x 49 Cartesian cells. The pollutant dispersion was simulated in a pseudo-transient mode using the flow data field modeled at every hour, grouped as if they were generated from an unsteady simulation, with a time step of 1h. The movies of the flow and pollution dispersion can be visualized from www.ascomp.ch. The IST technique was employed to map the buildings into a Cartesian mesh (the obstacles include the tall-rise building planned for construction neighboring the Blue-Win site).

The domain extension for all wind incidences was set horizontally as follows: Xmin = -505m and Xmax = 680m; Ymin = -308m and Ymax = 1260m. The vertical extension covered a large portion of the boundary layer; up to Zmax = 500m. The meteorological conditions were accounted for rigorously, in that the model distinguishes for instance between neutral
and thermally stable or unstable conditions (Garratt, 1992) both for the inflow and initial boundary conditions and for the k-epsilon modeling. Surface roughness was treated following the modified wall functions of (Lakehal, 1998; Lakehal, 1999). The concept requires reading the meteorological data from an open file, create the inflow boundary conditions, adjust automatically the grid according to the wind direction, store steady state flow fields, and finally read all these files and solve for the concentration in transient mode.

8.2 Simulation Results

Figure 25: Wind-acceleration contours at a specific date and time of the day, and isocountours of the time-averaged (over 3 days) comfort map for pedestrians.

The results shown in Fig. 24 depict the pollutant diffusion at one instant of the three-day simulation period. The color map on the ground indicates that the scalar has indeed deposited over the ground all around the area. A final output of the model simulation is to derive a probability density function (PDF) map, which shows what the direction the mostly exposed to the pollution is. This result which will depend on the period of time considered (e.g. one year time) could be used by the local authorities before taking the decision to deliver a building permission. A school or a kindergarten should not be placed in the direction where we have maximum PDF of pollution concentration. The same can be envisaged in a city where there exists a potential accidental release of toxic pollution from a chimney; the authorities could beforehand use a CFD tool like TransAT to calculate the flow and pollution over one year (a steady state simulation for every hour) using local meteorological data, store the results, estimate a PDF of concentration, determine the most exposed area of the city, and establish an emergency plan in case an accident based on the analysis.

Snapshots of the wind acceleration and comfort maps are shown in Fig. 25. The comfort map shown in the bottom panel is based on the model of Sanz-Andres and Cuerva (2006). It refers to comfortable for sitting areas as Label-(1) isocontours; comfortable for walking areas as Label-(2) isocontours; comfortable for Brisk walking areas as Label-(3) isocontours; and critical zones areas as Label-(4) isocontours. Comfortable zones for sitting areas (Label 1 isocontours) are identified in the Technopark buildings, represented as a double-opposed E-like structure on the plots, which was to be expected indeed.

9. Conclusions

The TransAT code using the IST technology has been validated for 3D turbulent flows past complex buildings, and pollution dispersion in the urban canopy. This is true for both RANS and scale-resolving strategies, LES and V-LES. The IST methodology, using the Google-Earth open software, has further been employed for a real situation involving pollution around an...
active area of the city of Zurich. Another novel modeling technique has been developed to analyze the pollution dispersion in the vicinity of a building under design using past meteorological data. The approach is based on pseudo-transient modeling, where steady-state flow solutions collected at each hour are used for transient advection of the scalar. Radiative heat transfer is also incorporated, including detailed sun rays radiation/reflection by the buildings inner and outer walls.

References


MUST experiment simulations using CFD and integral models R Buccolieri, SD Sabatino International Journal of Environment and Pollution 44 (1), 376-384


