e provision of information,
energy consultants in the
dings and urban planning.
f the relation climate-habitat-
design process, and not as a
sals.

g in general is very important
he country is employed in this

rder to prepare engineers and
utilization of natural energies

generate multiple actions, at

ur country due to a lack in the
field must be developed and

nust be put into action through
he various socio-cultural levels

s together with an "Usage and
you buy a new car. In this way
anitation, safety, maintenance,
al utilization of energy in the

sidered dynamic systems in
ment; these systems can be
r as participant, and the clear

Numerical Simulation of Indoor Environment in Large Indoor Spaces with Natural Ventilation

Part1: Development of a Simultaneous Simulation Technique for Inside and Outside Airflow of Large Indoor Stadiums

SUWA, Yoshihide* and SANO, Hitomi**

* Computational Eng.Dep.,Research Inst.of Thech.,OBAYASHI Corp.,640 Shimokiyoto 4-chome,Kiyose-shi,Tokyo 204,Japan **Phone:** +81 424(95)1269, **Facsimile:** +81 424(95)0906

** Architectural Eng.Sect.,Construction Dep.,Tokyo Electric Power Co.Inc.,9-5 Fujimi 2-chome, Chuo-ku,Chiba 260,Japan **Phone:** +81 43(224)3111, **Facsimile:** +81 43(221)5816

Abstract A purpose of this research work is to study the environmental control in large indoor stadiums which utilize the natural ventilation. In these problems, effects by the airflow in and around the stadium should be analyzed. In this research work, a numerical analysis technique which enables simultaneous simulation of indoor airflow and the airflow around buildings was developed adopting composite grid coordinate calculation technique. In this paper, an outline of this technique is described.

1. Introduction

There are several scales of phenomena in the flow field in and around buildings. Especially in problems concerned with the use of natural ventilation, both effects by inside and outside airflow of the building should be considered. In this research work, a new numerical analysis technique which enables simultaneous simulation of indoor airflow and the airflow around buildings was developed. Usually, difficulties on solving these kinds of problems are caused by the complexity of flow field configurations. In a present technique, flow fields are modeled in separated local unit grid systems as shown in Fig.1, and fundamental equations are solved using overset type composite grid coordinate calculation technique. Using this technique, even much complex flow field problems can be treated easily.

2. Outline of numerical formulations

2.1 Fundamental equations and discretisation

In this research work, incompressible viscous fluid flow was assumed. Adopting the generalized grid coordinate system (ξ, η, ζ), an unified form of fundamental equations is written as eq.(1).

$$J\phi_{,t} + \frac{\partial (JU_i\phi)}{\partial \xi_i} - \frac{\partial}{\partial \xi_i}\left\{\Gamma J\left(q_{ij}\frac{\partial \phi}{\partial \xi_j}\right)\right\} = J \cdot S_\phi \qquad \text{...(1)}$$

Here, the variable ϕ is an unified variable which represents ρu_i in momentum equation or k or ε in transport equations assumed in the standard k-ε type turbulence model. Discretised result of eq.(1) in a control volume defined as shown in Fig.2 is obtained as eq.(2).

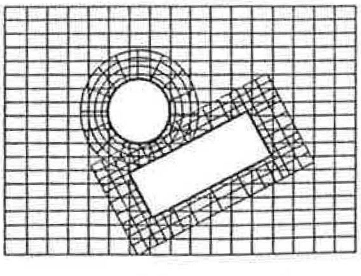


Fig.1 Overset type composite grid coordinate system

$$
\begin{aligned}
a_P\phi_P &= \sum_{nb} a_{nb}\phi_{nb} + b_\phi + J\cdot S_\phi \Delta\xi\Delta\eta\Delta\zeta \\
&= a_W\phi_W + a_E\phi_E + a_S\phi_S \\
&\quad + a_N\phi_N + a_B\phi_B + a_T\phi_T \\
&\quad + b_\phi + J\cdot\left(S^0 + S^1\phi\right)\Delta\xi\Delta\eta\Delta\zeta \quad ...(2)
\end{aligned}
$$

Adopting the under-relaxation factor α_ϕ, eq.(2) is re-written in following iterative form.

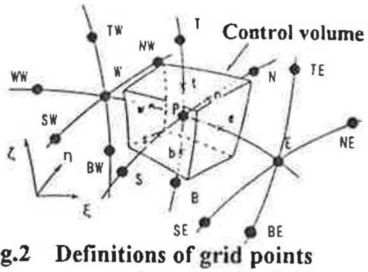


Fig.2 Definitions of grid points and a control volume

$$
\frac{a_P\phi_P^{m+1}}{\alpha_\phi} - \left(\sum_{nb} a_{nb}\phi_{nb} + b_\phi\right)^{m+1} = J\cdot S_\phi\Delta\xi\Delta\eta\Delta\zeta + \frac{1-\alpha_\phi}{\alpha_\phi} a_P\phi_p^m \quad ...(3)
$$

2.2 Linkage of variables in overlapped regions

In a present technique, simulated flow field is modeled as a composite system of different unit grid systems. Variables in each unit grid system is linked in overlapped regions. On this purpose, the fortified solution technique [Fujii,1995] was adopted. In this technique, variables in overlapped regions are combined using a weighting parameter χ which takes the value of $[0,\infty]$. Using this technique, eq.(3) is again re-written as follows.

$$
(1+\chi)\frac{a_P}{\alpha_\phi}{}^A\phi_P^{m+1} - {}^AF^{m+1} = {}^Ac_\phi + \chi\frac{a_P}{\alpha_\phi}{}^B\phi_P \quad ...(4)
$$

$$
F^{m+1} = \left(\sum_{nb} a_{nb}\phi_{nb} + b_\phi\right)^{m+1}, \quad c_\phi = J\cdot S_\phi\Delta\xi\Delta\eta\Delta\zeta + \frac{1-\alpha_\phi}{\alpha_\phi} a_P\phi_P^m
$$

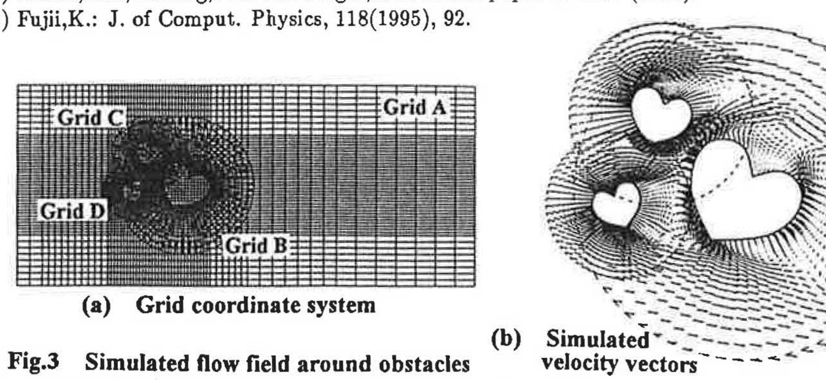
Eq.(4) is an iterative equation in grid A, which means that a variable ${}^B\phi_P$ in grid B is combined to the variable ${}^A\phi_P$ in grid A with weighting ratio of $\{\chi/(1+\chi)\}/\{1-\chi/(1+\chi)\}$. In this description, equation system is still symmetric, and it is possible to adopt simple matrix solving technique eg. as TDMA.

3. Applications

One of the applications of this technique is shown in Fig.3. Developed technique has several advantages, and it enables the airflow analysis in much complicated flow field problems which could not treated using previous any techniques. The other application for exact architectural problem is shown in part 2.

[References]

1) Benek,J.A., Buning,P.G. and Steger,J.L.: AIAA paper 85-1523 (1985).
2) Fujii,K.: J. of Comput. Physics, 118(1995), 92.



(a) Grid coordinate system

(b) Simulated velocity vectors

Fig.3 Simulated flow field around obstacles obtained using a present calculation technique

---

Numerical Si
in Large Indoor

Part2: Studies on char
openings and effe

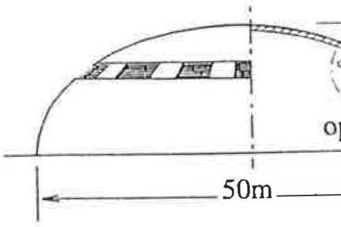
SANO, I

- Architectural Eng.Sect.,Const
Chuo-ku,Chiba 260,Japan F
-- Computational Eng.Dep.,Rese
4-chome,Kiyose-shi,Tokyo 204

Abstract It is necessary to cl
ventilation. Usually, openings usec
rain water, and sometimes they alsc
opening with complicated configur
technique, and the characteristics
separations generated at bents red
of draft airflow were dominated by

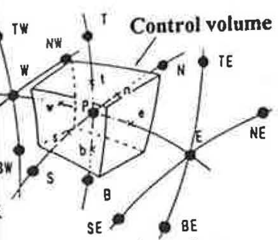
1. Introduction

Recently, several large indoo
huge amount of energy is necess
of the approach to reduce the e
sary to clarify the characteristi
most of previous research works
the hydrostatic pressure theory.
complicated configuration. In th
were studied using the numerica



(a) Stadium

Fig.1 Simulated opening co



finitions of grid points
and a control volume

$$-\alpha_\phi \over \alpha_\phi}a_P\phi_p^m \quad ...(3)$$

s a composite system of different ed in overlapped regions. On this pted. In this technique, variables meter χ which takes the value of ows.

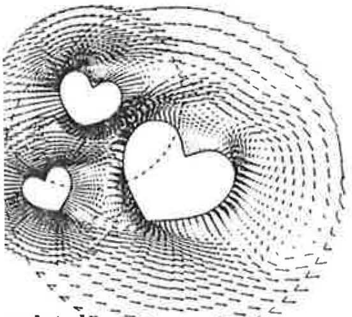
...(4)

$$+\frac{1-\alpha_\phi}{\alpha_\phi}a_P\phi_P^m$$

variable ${}^B\phi_P$ in grid B is combined $1+\chi)\}/\{1-\chi/(1+\chi)\}$. In this ble to adopt simple matrix solving

3. Developed technique has several plicated flow field problems which application for exact architectural

1523 (1985).



nulated
ocity vectors

Numerical Simulation of Indoor Environment in Large Indoor Spaces with Natural Ventilation

Part2: Studies on characteristics of draft airflow through complicated openings and effects of natural ventilation on indoor environments

SANO, Hitomi* and SUWA, Yoshihide**

* Architectural Eng.Sect.,Construction Dep.,Tokyo Electric Power Co.Inc.,9-5 Fujimi 2-chome,
Chuo-ku,Chiba 260,Japan **Phone:** +81 43(224)3111, **Facsimile:** +81 43(221)5816

** Computational Eng.Dep.,Research Inst.of Thech.,OBAYASHI Corp.,640 Shimokiyoto
4-chome,Kiyose-shi,Tokyo 204,Japan **Phone:** +81 424(95)1269, **Facsimile:** +81 424(95)0906

Abstract It is necessary to clarify the characteristics of openings to study the effect of natural ventilation. Usually, openings used for large indoor stadiums have steep bents to avoid entering of rain water, and sometimes they also have grilles and baffles. In this research work, airflow through the opening with complicated configuration was simulated using a composite grid coordinate calculation technique, and the characteristics were studied. As the result of the study, it was found that flow separations generated at bents reduced the width of main stream in the opening, and characteristics of draft airflow were dominated by these separations.

1. Introduction

Recently, several large indoor stadiums are constructed. In these large indoor spaces, a huge amount of energy is necessary for air conditioning. An use of natural ventilation is one of the approach to reduce the energy. To study the effect of natural ventilation, it is necessary to clarify the characteristics of openings which introduce the outside air. However, in most of previous research works, characteristics of these openings has been studied based on the hydrostatic pressure theory. So, it was impossible to apply these results on openings with complicated configuration. In this research work, draft airflow characteristics of these openings were studied using the numerical simulation technique developed in part 1.

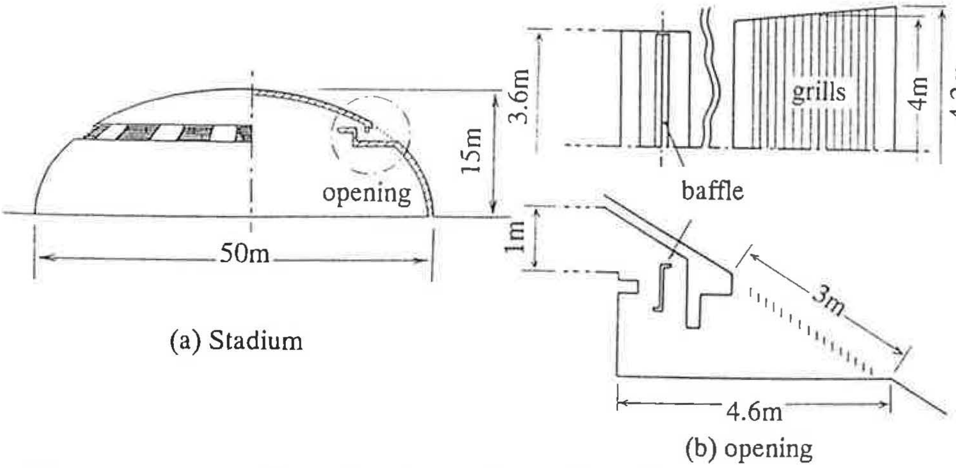


Fig.1 Simulated opening configuration and the stadium which the openings are installed

2. Outline of numerical analysis technique

Openings simulated in this study have complicated configuration, and both inside and outside airflow states affect on the draft airflow. Here, the numerical simulation technique developed in part 1 was adopted to study the draft airflow through openings. In this technique, flow field is separated in parts and the airflow is simulated using the composite grid coordinate system. So, it enables the simulation in flow fields wirh complicated configurations and also simultaneous simulation of inside and outside airflow. Effect by the airflow turbulence was considered using the standard k-ε model.

3. Analyzed opening configuration and its modeling

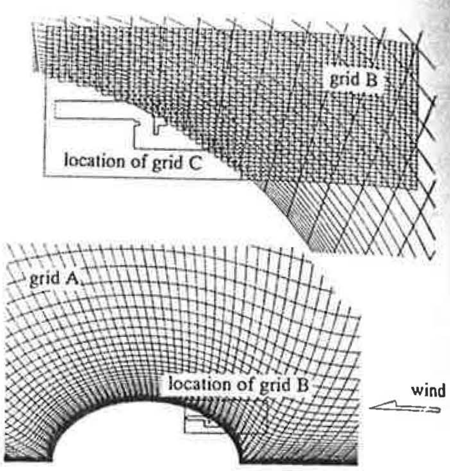
Fig.1 shows configurations of analyzed opening and the stadium which the openings are installed. The stadium has a dimension of 50 meters in diameter and 15 meters in height. Analyzed opening has an area of inlet 3×4 meters, and it has two steep bends to avoid entering of rain water. In this study, some other variations of openings which has grills and baffles were simulated.

Flow field around the stadium and inside the opening were modeled in the grid coordinate system shown in Fig.2. The airflow around the stadium was simulated using grid A, and the result was used to setup the boundary conditions of grid B. Grids B to F were linked using the overset type composite grid coordinate calculation technique, and the draft airflow through the opening was simulated.

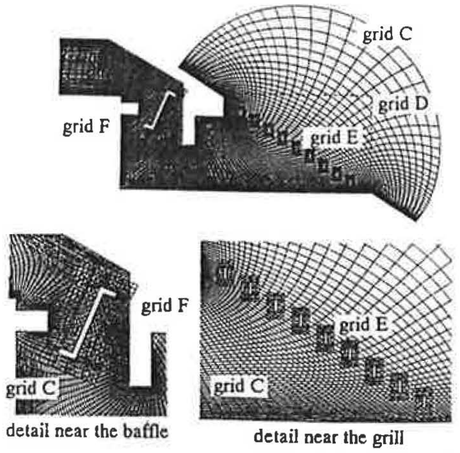
4. Simulated results and considerations

4.1 Airflow state in the opening

Some of the results of the simulation is shown in Fig.3. These figures show the velocity and pressure fields in the opening at Reynolds number 10^3. In these results flow separations are observed around grills and at bends. Especially, the separation generated at bending edge is very large in each case, and it reduces the width of the main stream. Draft airflow characteristics depend on the pressure drop at choked point of the main stream. As the result of the simulation, it was found that a dominant effect on the pressure drop in this kind of openings was separations generated at the bending edges. Same kinds of effects were

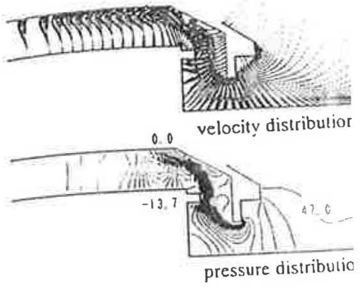


(a) Grid system used for airflow analysis around the stadium

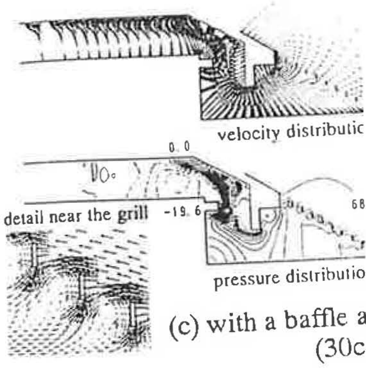


(b) Grid system used for airflow analysis through the opening

Fig.2 Grid system used to calculate draft airflow through the opening



(a) with bends



(c) with a baffle a
(30c

Fig.3 Simulated re

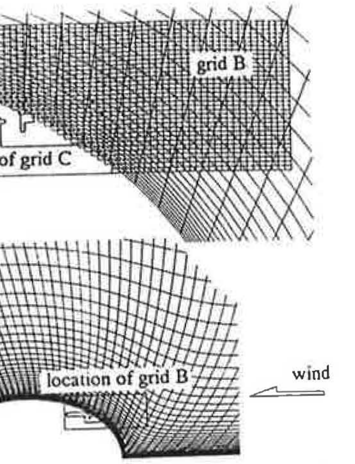
also observed in other variations
calculated from these simulation
of simple openings.

4.2 Change of discharge coef

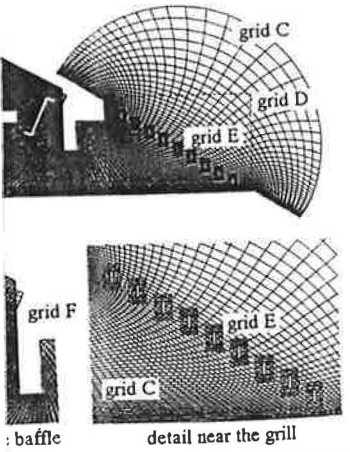
Fig.4 shows the change of disc
constant during higher Reynolds
Reynolds number region. This cu
at bents, and it depends on the g
the steep increase in discharge c
dissipation of the separations.

4.3 Effect of baffle choke an

Fig.5 shows the difference of t
tern at downstream of bending
and it changes to confused wake

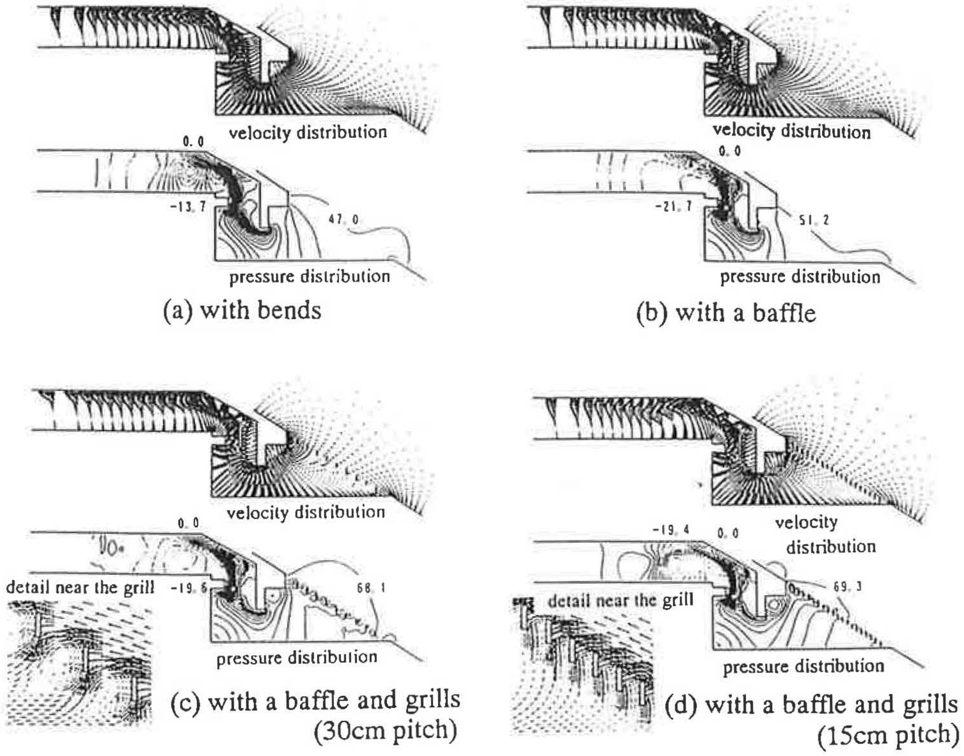


stem used for airflow analysis
around the stadium



ystem used for airflow analysis
through the opening

id system used to calculate
ft airflow through the opening



(a) with bends

(b) with a baffle

(c) with a baffle and grills (30cm pitch)

(d) with a baffle and grills (15cm pitch)

Fig.3 Simulated result of draft airflow through the opening

also observed in other variations which did not installed grills or baffles. Discharge coefficients calculated from these simulation results were 0.2 to 0.3 and were much lower than the values of simple openings.

4.2 Change of discharge coefficient at different Reynolds number

Fig.4 shows the change of discharge coefficient vs. Reynolds number. Discharge coefficient is constant during higher Reynolds number than specific value, and it increases steeply at lower Reynolds number region. This curve coincides with the change of the main stream section area at bents, and it depends on the growth or dissipation of the separation. It was recognized that the steep increase in discharge coefficient at lower Reynolds number region was caused by the dissipation of the separations.

4.3 Effect of baffle choke angle

Fig.5 shows the difference of the airflow when the baffle choke angle was changed. Flow pattern at downstream of bending edges shows a drastic change in accordance with baffle angle, and it changes to confused wake patterns.

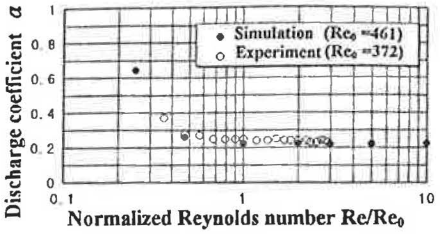
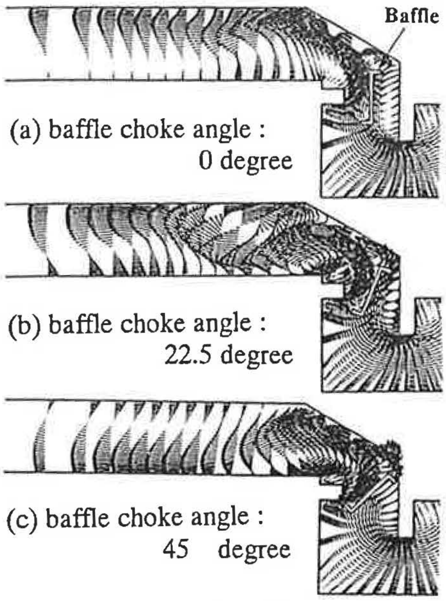


Fig.4 Change of discharge coefficient vs. Reynolds number



(a) baffle choke angle : 0 degree

(b) baffle choke angle : 22.5 degree

(c) baffle choke angle : 45 degree

Fig.5 Change of flow field in the opening when baffle choke angle was changed

Discharge coefficient shows a steep change when the baffle choke angle is between 30 to 60 degree. In these cases, it also seems that change of the flow pattern and generation of the separation affect on the draft airflow characteristics.

5. Conclusion

Airflow through the opening with complicated configuration was simulated using a composite grid coordinate calculation technique, and the characteristics were studied. As the result of the study, it was found that flow separations generated at bents reduced the width of main stream in the opening, and characteristics of the draft airflow in this kind of openings were dominated by these separations.

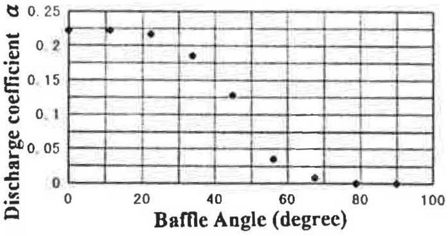


Fig.6 Change of discharge coefficient vs. baffle choke angle

THE AUSTRALASIAN

John A. Ba

The

ABSTRACT The Australa energy rating scheme based of New South Wales. The pr Government to provide a sim identify the most energy effic work was completed in 1995 September 1996. This paper the scheme has been develop

1.0 OVERVIEW

1996 has seen the introducti residential windows. The sc Council (AWC) by Solarch house energy rating is descri has the backing of the wind utilities.

- WERS enables residential comfort impact on a whole to that already used for re for energy efficiency; WE for New Zealand, it wi independent of any one m credible system for testing
- To participate in WERS, their windows must meet the quality assurance th performance and is desi House Energy Rating Sc with the benefit of co