RESULTS FROM A CFD REFERENCE STUDY INTO THE MODELLING OF HEAT AND SMOKE TRANSPORT BY DIFFERENT CFD-PRACTITIONERS

Marcel Loomans[1], Tony Lemaire[2], and Mirjam van der Plas[3]
[1]Eindhoven University of Technology, Eindhoven, The Netherlands
[2]Efectis, Rijswijk, The Netherlands
[3]NIFV, Arnhem, The Netherlands

ABSTRACT

The paper describes results from a reference study that focuses on the application of the Computational Fluid Dynamics (CFD-) technique for heat and smoke transport in practice. Goal of the study is to obtain insight into the amount and causes of the spread of CFD-results when applied by different users.

In this study several CFD-practitioners have solved the same relatively well described flow problem. The obtained results have been compared. They show a clear spread which to some extent can be explained by the assumptions made for the modelling and solving of the problem. Participants adhering closest to proposed problem best practice obtain similar results, independent of the CFD code applied. Another conclusion is that a good overview of the model interpretations for the flow problem adds to the value of the results. However, than it is also important that those who need to assess the results can value this overview.

INTRODUCTION

Application of the Computational Fluid Dynamics (CFD-) technique for design problems in practice is getting more widespread. In the Netherlands this specifically deals with equivalence of performance related to heat and smoke transport in case of fire for non standard design solutions. CFD than is used to support applicability of the design solution. Given the complexity of this type of flow problems and the fact that experimental validation generally is not possible, validity of the numerical results not always is clear. Besides the controlling authorities very often do not have the required knowledge to assess validity of such CFD-results (Kobes et al. 2006).

Therefore, a so-called reference study has been initiated. The main goal of this study is to obtain insight in the amount and background on the spread of CFD-results when the same flow problem is modelled and solved by different CFD users. Furthermore, the wish is to arrive at (minimum) recommendations for performing CFD-studies for similar type of flow problems, as well for the executing as assessing parties.

Participants in the study are consultants (relative novice to experienced) that use CFD in their daily consultancy practice.

In the study in total three cases will be investigated, with a focus on the discretization of the problem including the applied physical submodels (for given well defined boundary conditions). The first case has been kept as relatively simple as possible. The following cases will show an increasing complexity. In all cases the flow problem is focused on heat and smoke transport for specific application areas such as car parks and atria.

In this paper a summary is given of the results from the first case.

METHOD

Procedure

In the study a description is given of the flow problem that is to be investigated (case). Following this description, participants (voluntary) performed simulations of this case. They were given approximately 6 weeks to perform the calculations and send in the results. All received information then was dealt with anonymously.

The results contained general information on the applied CFD-code, application area and experience of the participant for simulating this type of flow problems. Furthermore, specific information was asked for with respect to the modelling of the case to be investigated. This consisted of the discretization of the geometry, applied submodels and solving method. Finally, for the positions where measurement data was available, numerical data was asked for. Contour plots of the temperature field and vector plots of the velocity field completed the required information.

Comparison of the results mainly has been performed based on the choices made for several aspects of the CFD modelling in relation to the numerical data obtained. For example, how radiation heat transfer has been dealt with.

Case description

The point-of-departure for the first case was to start with a situation that was as simple as possible, but nevertheless representative with respect to heat and smoke transport. This was translated in a case that could be assessed two dimensional (2D), a problem that can be regarded stationary and with a relatively simple and clear geometry.

With the above considerations the so-called Heselden case was chosen as the first case. In this case in 1971 Heselden performed fire experiments in a shopping mall that was close to demolition (Heselden 1972). The experiments have been performed such that a (as good as possible) 2D stationary flow situation existed. In literature the case has been described in an article by Markatos and Malin (1982). This reference was used for the description of the case.

The flow problem is a room (L×W×H = 9×6×3m) in which a fire source has been placed on the floor over the full width of the room (6 m) at a distance of 1.25 m from the back wall (middle; size fire source: 0.5 m). In the front wall of the room an opening is positioned that allows exchange of air with the outdoor environment. A plan of the room is presented in Figure 1. The side walls continue into the outdoor area, after the soffit. In Figure 2 additional information is given on the geometry of the outdoor area (x >9 m). The size of the outdoor area to be modelled has been left open for interpretation by the participants in the study.

In the experimental set-up an opening was present over the full width of the room. A soffit from the ceiling restricted the opening height to 2.0 m (height soffit: 1.0 m). The applied fuel in the experiments was 'industrialized methylated spirit' with a heat release of 2.04 MW. This heat release was based on the loss of weight of the fuel. The flames reached up to approximately half the height of the room. The outdoor temperature was prescribed at 293 K. The temperatures for the floor, ceiling and walls were allowed to be kept at a constant temperature similar to the outdoor temperature.

In the experiments measurements have been performed at different positions. The air temperature has been measured at the positions I, J and K (see Figure 1), the air velocity at one position (triangle; Figure 1). The measurements have been performed by thermocouples and water cooled anemometers. All measurements have been performed in the centre line of the room at different distances from the fire (three stands with 9 thermocouples, with a vertical distance of 0.3 m, situated at 2.56 m, 5.76 m and 8.96 m from the back wall; one stand with 8 anemometers, with a vertical distance of 0.45 m, situated at 5.46 m from the back wall).

The accuracy of the measurements is relatively low. Radiation at the thermocouples near the fire can lead to a significant overestimation. Markatos mentions in the comparison of the results a value of 50 K for the lower measurement positions at x = 2.56 m. The air velocity sensors register the horizontal component starting from a velocity of approximately 0.5 m/s. A 1-to-1 agreement of the simulated data with the measured data therefore cannot (and should not) be strived for. The data however do give an indication of the flow field in the room.

In the first description of the case no information was provided on the radiant part of the fire. However, based on the information with respect to the type of fuel it was not possible to derive the amount of radiation. Because of this reference was made to Markatos et al. (1982), who assume a radiation fraction of 20%. From this one may conclude that in this case the importance of radiation in the total heat transfer was lower than usually present in practice. This is mainly related to the type of fuel applied.

RESULTS

Participants, CFD code

In total, results have been obtained from 12 participants from 11 companies. They investigated the above described Heselden case with CFD. Most participants had more than 3 years experience in the use of CFD. Within the companies CFD application was mainly focused at fire safety engineering.

In this study a clear distinction can be made in the type of CFD-packages that have been used. On the one side, these are commercial CFD-codes, mainly based on RANS[1]-techniques for modelling turbulence. In this case Phoenics was applied most. On the other side Fire Dynamics Simulator (FDS) was used. This code is available free of charge at NIST (2007). FDS applies the LES[2]-technique for modelling turbulence. In this study it was intended to have information available from at least three different CFD-codes. For this case in total four different CFD-codes have been used.

Problem definition and translation into a model

The participants have treated several aspects of the modelling of the flow problem in a different way. Below, some of these aspects will be explained. A more extended description of the other aspects is given in Loomans et al. (2008). In the discussion of the results also these aspects will be dealt with.

With respect to the geometry, variation mainly is found in the modelling of the outdoor space. Goal of the modelling of this space is to allow development of the flow field near the opening of the room as independent as possible from the boundary conditions. A division in the results from the participants is possible between situations where the outdoor space is not modelled at all or just as high as the fire room height and situations where a higher outdoor space is modelled together with the fire room.

The effect of this division on the numerical results is visible in the graphs shown in Figure 3. In this case a selection is made of the available datasets of the participants. In this selection datasets are grouped

[1] RANS: Reynolds-Averaging Navier Stokes
[2] LES: Large Eddy Simulation

that either did model the outdoor area as a relatively high and large outdoor space, or didn't model the outdoor space or modelled an outdoor space with a similar height as the fire room.

In the legend of each graph the number of datasets that are part of the individual selections is given between brackets. The separate datasets are presented with their average values and the overall minimum and maximum values (this means that these lines do not by definition represent a complete dataset from an individual participant). The thick line is the average value, the thin line with the same color and style represents the minimum/maximum value. In all graphs the average of all available datasets and the measurement values are shown for reference.

In the results different options of modelling the fire appear. A clear distinction can be found between the so-called Volumetric Heat Source approach and by modelling the fire at a (low block shaped) surface area (see Figure 4). For the Volumetric Heat Source approach information on the height of the source was provided for. The results show that this information not always was used. In some cases a combustion model was used. These datasets are not part of the above comparison.

Solver methodology

Important parts when applying CFD are the discretization of the problem, the applied turbulence model and other submodels.

Some of these aspects will be explained shortly. A more extended description of the other aspects is given in Loomans et al. (2008). In the discussion of the results these aspects will be dealt with as well.

The turbulence modelling is related to the applied CFD-code. FDS applies the LES technique; the other codes have RANS as basis (though some do allow the use of LES as an option for turbulence modelling). In case of RANS, all participants applied the k-ε model or a directly related model. Figure 5 presents the comparison for this topic.

Radiation forms an important aspect in the heat balance for a fire. In the calculation radiation can be accounted for in different ways. The most detailed approach uses a separate radiation model. Instead of use of such a model, the effect of radiation can be accounted for by correcting the total heat release. Then no radiation model is applied, but for the heat release of the fire only the convective part is defined. Figure 6 presents a comparison where no correction has been made for the effect radiation has on the heat release (i.e. no radiation model or application of the convective part), and results where the radiant part has been subtracted from the overall value for the heat release. In both cases a radiation model is not used.

DISCUSSION

This paragraph presents a short discussion of the above presented results. It then will continue with a more general judgement of the results.

Comparison of individual aspects

The majority of the participants has modelled a high outdoor space in the flow problem. However, when the results are compared, the effect of this high space appears limited. From the comparison no clear distinction can be found between the modelling or not modelling (or low height) of the outdoor space.

Point-of-departure for the reference study was to focus mainly on the modelling of heat and smoke transport, not on the modelling of the fire itself (combustion modelling). The actually available information for the fire in the experiments for this case was limited. The Heselden case is a situation (i.e. relatively small room) for which the given fire description perhaps was less well valid. The fire should have been defined better to allow a better comparison with the measurement data. As indicated, the available information on the fire was restricted and incomplete. This meant that modelling the fire (combustion) on the basis of the available information was not well possible. No information was provided on the smoke release, so this is not taken into account in this case as well.

The effect of the way the heat release has been modelled (volume versus surface) shows small differences. This translates mainly into the temperature stratification, i.e. the presence of a clear high temperature layer (compare to smoke layer). In case the volumetric heat source approach is applied the division between the two layers is sharper. A remark however should be made that the applied fire volumes (height) differed between participants.

Comparison of the results for the different turbulence models showed nearly no differences in the average velocities. The temperature in the smoke layer for the LES results however is approximately 50 K lower than for the datasets that have been simulated with a RANS model. The explanation for this difference most probably should not be found in the turbulence modelling, but in the modelling of the fire.

A reduced heat release, when the radiant heat transfer is corrected for in the convective heat release, normally would result in lower air temperatures in the smoke layer. However, in this case also the fire induced flow will be reduced, resulting in a lower cooling capacity that is induced into the fire room. As a result the average air temperature in the smoke layer will be higher. The results shown in Figure 6 appear to support this explanation. The temperature profiles indicate relatively small differences for the cases with or without corection of the convective heat release for radiation. For the velocity profile higher velocities are found near floor and ceiling level when radiation is not accounted for (i.e. a higher heat release). Correcting for the radiation part improves the agreement with the measured velocities.

Best practice

The above discussion presented some examples of the comparison of the datasets for individual aspects of the modelling of the flow problem. A more specified comparison or comparison of combined aspects was assumed less sensible because of the number of datasets available. If a selection would have been made on more than one aspect, the number of applicable datasets would have been too limited.

Instead, a definition for a 'best practice' was developed that was assumed sensible for the flow problem under investigation. In this 'best practice' for the individual aspects the preferred modelling approach was indicated. The individual datasets were rated against this. Table 1 presents a summary of the 'best practice' assumed for this case. Besides experience of the authors, e.g., also information from Gobeau et al. (2002) was used. Remark that sometimes, based on efficiency arguments, it is not always possible to apply all 'best practice' aspects. However, in that case it is important to be able to assess the consequences of this.

For each available dataset an assessment is given for the different aspects as indicated in Table 1. In total a maximum of nine (combined) aspects could be assessed for each dataset. Partly an extra weighting was given for specific choices. This allowed a maximum of 12 points to be gained. A further explanation to this is given in Loomans et al. (2008).

The datasets that adhere best and least to the 'best practice' are determined by the amount of points from the assessment, combined with the number of aspects which were respected by the participant for this dataset ('aspects valued'). In addition to the overall best practice also a specific selection has been made of the datasets that have been obtained with RANS-models. In Table 2 a summary is given of both assessments. Figure 7 graphically presents the results for the selected datasets for the overall best practice

The agreement in the datasets for the best three overall best practice shows that different users are able to obtain similar results for the defined flow problem. In comparison to the least three datasets a clear and sharp smoke layer is calculated. This agrees with the course of the measured temperatures. The best three results were obtained with three different CFD codes.

Nevertheless, it is notable that between the three best datasets the difference in the smoke layer temperature increases at larger distance from the fire source (position J, K [not shown]). This difference is mainly explained by the difference in modelling the radiant heat transfer. The two datasets with a lower smoke layer temperature at position J and K apply a radiation model to include the effect of radiation. In the other dataset radiation is corrected for in the convective part of the heat release. Besides, for this dataset the boundary condition for the wall deviated from the others. In this case use was made of an external temperature and heat transfer coefficient, which will result in a lower heat loss at the walls.

The least three datasets show a larger spread, as well in the temperature profiles as in the velocity profiles. Close to the fire for these datasets no clear (sharp) smoke layer is present and there is little agreement with the measured temperature profile. For two of the three datasets, the fire was modelled as a heated surface instead of a volumetric heat source. This suggests for the use of a volumetric heat source approach for this case.

CONCLUSION

The results and discussion indicate that it is possible to obtain similar results from different users of a CFD code for a (relatively) detailed description of a flow problem. The applied code is relatively independent of that. Remarkable is that despite the comparable best practice approach, still significant differences appear. This relates to the difference in interpretation of parts of the defined flow problem.

In current practice when problems generally are more complex these type of differences in interpretation most probably will take an even more important position. Then it is important to have knowledge of the points-of-departure that relate to the CFD results and the assumptions that have been made. An overview of simulation details and choices as indicated in Appendix D of a Dutch Design standard (Ontwerp NEN 6098 (NEN, 2007)) would adhere to this. It nevertheless also remains important that others who apply and assess the results from such a CFD study are aware of these details and choices and, to some extent, can value the effect of them.

This reference study has been continued with a second more complex case.

ACKNOWLEDGEMENT

This research would not have been possible without to voluntary cooperation of the participants and their employers that allowed them to participate in this study.

REFERENCES

Gobeau, N., Ledin, H.S., Lea, C.J. 2002 Guidance for HSE Inspectors: Smoke movement in complex enclosed spaces - Assessment of Computational Fluid Dynamics. HSL/2002/29. Health & Safety Laboratory. Fire and Explosion Group. Buxton. United Kingdom.

Heselden, A. J. M. 1972. Fire problems of pedestrian precincts. Part 1. The smoke production of various materials. Joint Fire Research Organisation, Fire Research Station. Fire Research Note 856.

Kobes, M. , Rosmuller, N., Schokker, J.J., Vliet, V.M.P. van. 2006. Verkenning van simulatiemodellen: Brand- en rookontwikkeling, evacuatie- en interventiemodellering. Versie 442N6001/30-08-06. Nederlands Instituut Fysieke Veiligheid Nibra, Arnhem, Nederland.

Loomans, M., Lemaire, A.D. , Plas, M. v.d. 2008. referentiestudie NIFV | IBPSA-NVL | TU/e - Deelrapportage Resultaten 1e casus (Heselden casus), NIFV, Arnhem, Nederland.

Markatos, N. C., Malin, M. R., Cox, G. 1982. Mathematical Modelling Of Buoyancy-Induced Smoke Flow In Enclosures, Int. J. Heat Mass Transfer, Vol 25 No.1, Pp.63-75.

NEN. 2007. Ontwerp NEN 6098 – Rookbeheersingssystemen voor mechanisch geventileerde parkeergarages. Versie uitsluitend voor commentaar. Nederlands Normalisatie-instituut, Delft.

NIST. 2007. Fire Dynamics Simulator (Version 5) User’s Guide. NIST Special Publication 1019-5. October 2007. National Institute of Standards and Technology, Washington DC, USA.

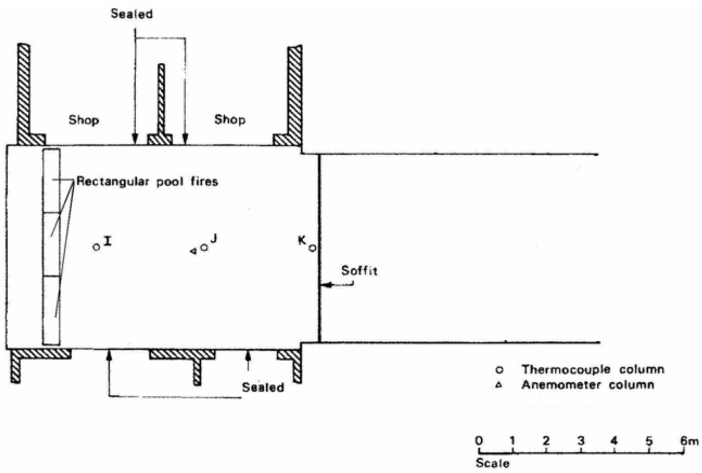
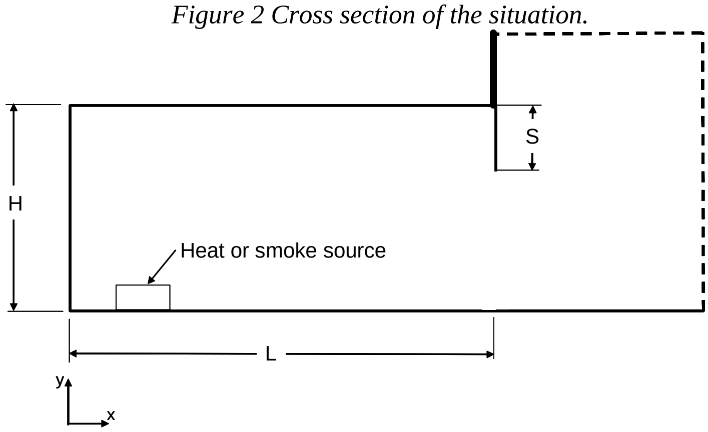


Figure 1 Plan of the situation (adapted from Markatos and Malin 1982).

Figure 2 Cross section of the situation.



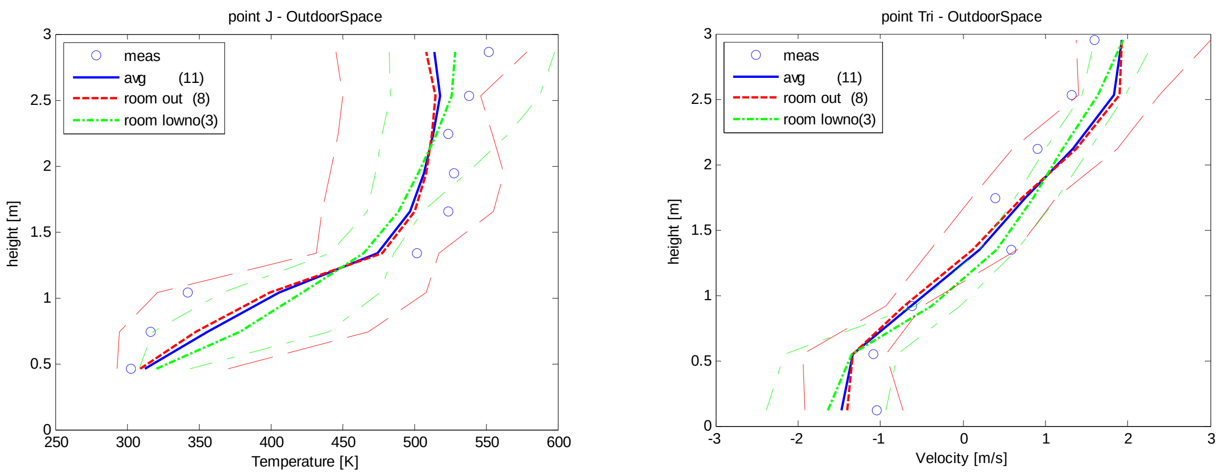


Figure 3 Difference in modelling a high outdoor space compared to modelling without such a space or a low outdoor space (for point J and Δ respectively).

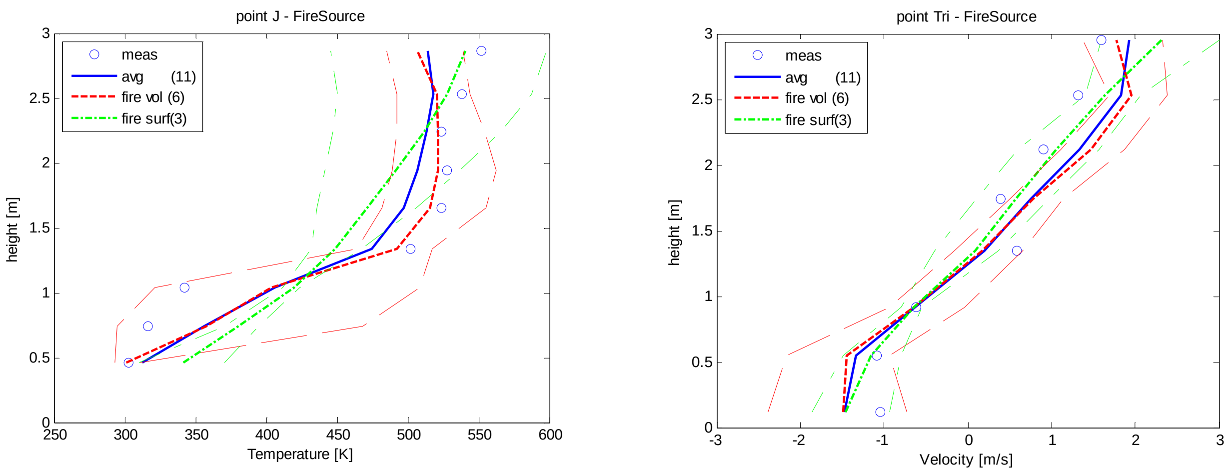


Figure 4 Difference in modelling of a fire; volumetric heat source approach compared to heat release at a surface.

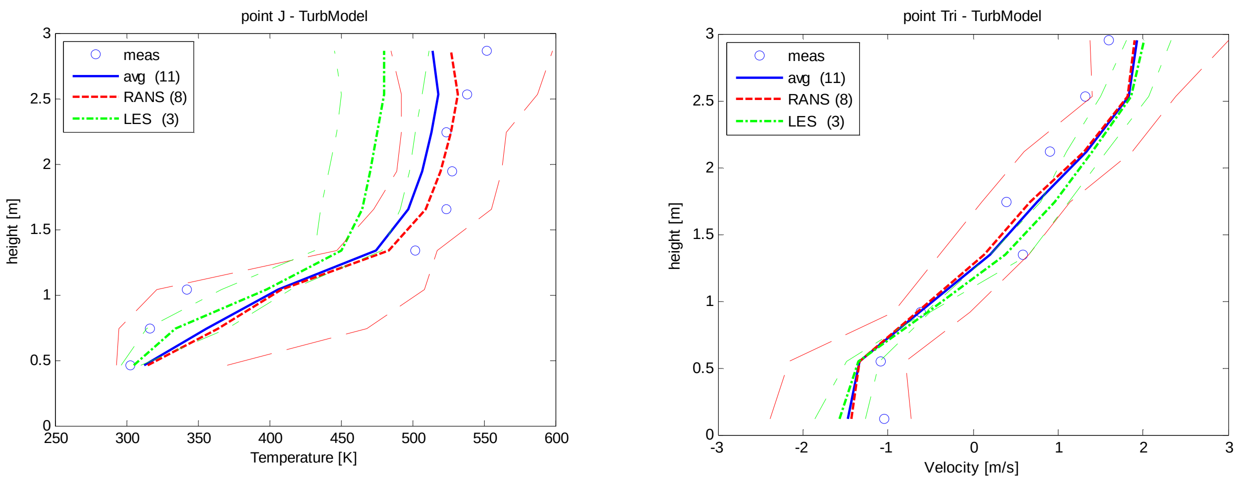


Figure 5 Difference in turbulence modelling; RANS compared to LES.

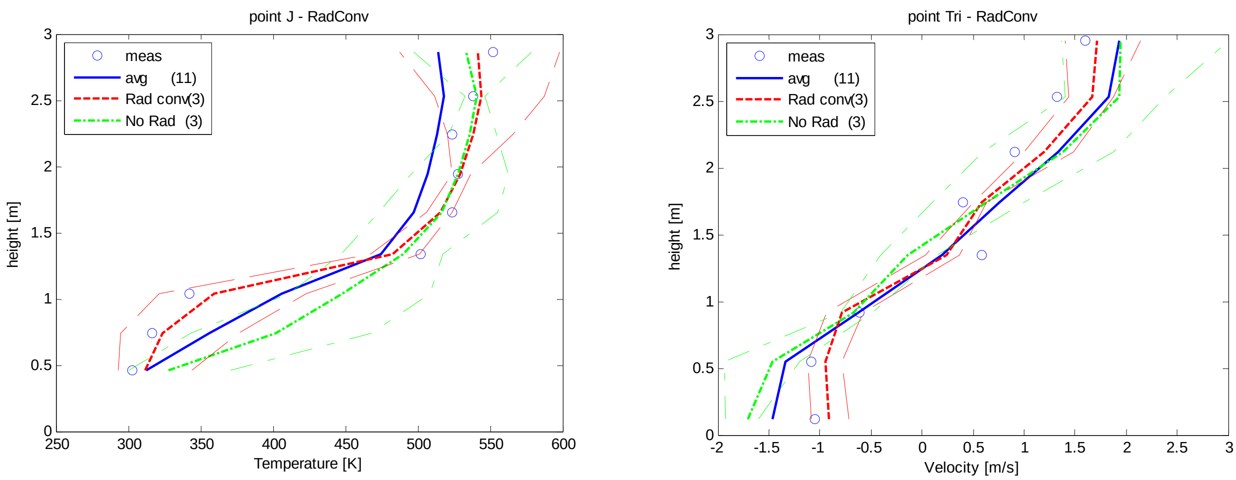


Figure 6 Difference in including radiant heat transfer; radiation taken into account in the convective heat source compared to neglecting the radiant part completely.

Table 1
Choices best practice for described case.

Aspect	Choice
Fire	A combustion model would have been the best choice for this relatively small room. Instead a 'volumetric heat source' approach would be a good choice as well. This approach is specifically useful if interest is highest in results further away from the fire source.
Outdoor space	Modelling of a high and sufficiently large outdoor space prohibits that the flow into the fire room is affected by the applied boundary conditions.
Boundary conditions wall	Based on the prescribed conditions adiabatic boundary conditions certainly should not be used.
Boundary conditions opening	A pressure boundary condition at sufficient distance from the opening of the fire room presents the most realistic situation for the given flow problem.
Grid	The sketched situation is 2D and may be modelled as such if the applied CFD-code is trusted for this. A 3D approach answers questions on this. The minimum grid size normally should be confirmed with a grid study, unless earlier similar studies resulted in clear guidelines with respect to the grid size. A minimum amount of grid cells was assumed to be required.
Turbulence	As well RANS as LES calculate similar results for the investigated flow problem. Underestimation of the smoke layer temperature for LES is mainly related to the fire modelling and radiation modelling applied.
Radiation	Radiation modelling for this case was not unambiguous as information on the fire was not complete. Correction of the convective heat release in that case is a good alternative, certainly when interest in the flow field is away from the fire source.
Solver procedure	To minimize numerical diffusion preferably use should be made of higher order discretization schemes. This is also related to the grid applied. Disadvantage of such schemes is that the solver process can be less stable.

Table 2
Summary of assessment of the available datasets in comparison to best practice.

Best practice	# points	# aspects valued	# aspects information unknown
Best (Overall)	9-11	8	0
Best (RANS)	9-10	7-8	0
Least	5-6	4-5	0-1

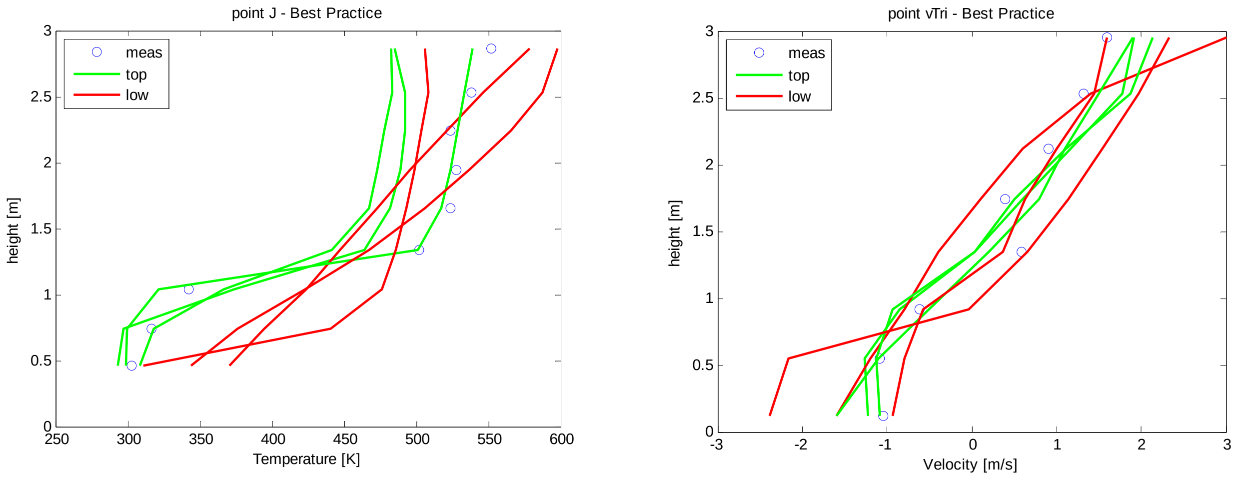


Figure 7. Comparison of datasets assessed according to best practice ('top' is best [overall] and' low' is least [overall]).