Automated two-way CFD fire-FEM thermo-mechanical coupling for global modelling of building structures under fire

Dr. Ir. H. Hofmeyer, Ir. J.A. Feenstra
(Eindhoven University of Technology TU/e, The Netherlands);

Ir. R.A.P. van Herpen
(TU/e and Nieman Raadgevende Ingenieurs, The Netherlands);

Prof. Dr. M. Mahendran
(Queensland University of Technology, Brisbane, Australia).

Abstract

For the global modelling of building structures, that is, the modelling of the overall topology of the structure, not limited to certain connections or components, the coupling of CFD fire simulations and FE thermo-mechanical models can be carried out by either one-way coupling or two-way coupling. Unlike one-way coupling, two-way coupling considers the effect of the structural response on the fire propagation, an effect that so far has only been studied for a very limited number of cases. The first part of this paper investigates the feasibility of two-way coupling. Its components, fire simulation, heat transfer analysis, and structural response analysis are introduced first. These components are coupled by three steps: (i) coupling of fire simulations to heat transfer analysis; (ii) coupling of heat transfer analysis to structural analysis; and (iii) coupling of structural response to fire simulation, in which the latter is unique for two-way coupling. Then the implementations of these couplings in C++ and Python are described in this paper. Finally, the setup of a master program in C++ is explained to automatically control the couplings and the programs of the components (FDS for fire simulation and Abaqus for heat transfer and structural analyses). Using this implementation, the second part of this paper shows the practical differences between one-way and two-way coupled analyses for an office space in fire that results in failure propagation of its thin-walled steel building façade. Although the results differ slightly for each of the several simulations due to random effects in the fire simulation, overall results are quite comparable. It can then be concluded that the implementation and the significant difference in failure progression of the facade illustrate the feasibility and the effectiveness of two-way coupling, respectively. However, further research to develop more advanced fire and structural models and validating them using experiments is required for an all-conclusive answer.

1. Introduction

As the built environment evolves, new challenges arise for researchers and fire fighters. Among others, the growing complexity of architectural designs, structural optimisation, and the use of innovative building materials and construction techniques yield structural behaviour exposed to a fire yet unknown, and possibly different from the current situation. Hence a better understanding of fires under these modern conditions, and means for predicting the temperature and smoke development involved, are crucial to both structural integrity and human safety under fire conditions.

The traditional approach for including a fire load in structural engineering is to impose prescriptive time-temperature curves, considered to be representative of a fire, onto a structural element, and to analyse the element's mechanical behaviour. Subsequently, the actual safety check revolves around meeting a certain period for which the component should resist the fire (European Committee for Standardization, 2012). In this traditional approach, the prescriptive time-temperature curve may not accurately model the randomness of the fire and additionally, it will not take into account the 3D geometrical relationships between the fire compartment and the structural system.

Therefore, recent advancements in structural fire analysis utilise advanced numerical methods to both simulate the fire and analyse the structure's thermal and structural behaviour (Duthinh *et al.*, 2008). More specifically, this involves Computational Fluid Dynamics (CFD) to model the fire driven fluid (i.e. gas) flow and to couple this data to a thermal and structural Finite Element Method (FEM) based analysis of the structural system under consideration. The approach of combining CFD and FEM analyses is commonly referred to as "coupled fire to thermo-mechanical analysis or "coupled CFD-FEM". Note that the resurgence of interest in the behaviour of structures under fire and the development of coupled approaches has partly been triggered by the unfortunate collapse of the World Trade Centre Towers in 2001. As a result, the National Construction Safety Team advises to enhance the capability of available computational software to study realistic fire behaviour, to analyse building response to fire, and to assist in the design of new fire protection systems (National Institute of Standards and Technology, 2005).

Although the coupling of CFD and FEM analyses thus seems to have particular benefits, it is not a trivial task as challenges are found in the underlying differences between CFD and FEM, e.g. with respect to the discretization, algorithms, and time scales. Research has been carried out to investigate and solve these challenges: Prasad and Baum (2005) developed an interface model, named "Fire Structural Interface" (FSI), which maps thermal boundary conditions from a so-called Fire Dynamic Simulator (FDS) (McGrattan *et al.*, 2013) to the heating analyses of complex structures by FE package Ansys. Later research on coupled CFD-FEM analyses by Baum (2011) discusses the role of uncertainty in input parameters, and the challenges found due to differences in spatial and temporal length scales, different numerical techniques; complexity of computer codes and the required computational resources. Baum (2011) also underlines the importance of the conjugate development of coupled CFD-FEM models for the quantitative assessment of fire effects on structures.

A coupled CFD-FEM analysis consists of three components: (a) a fire simulation, (b) a heat transfer analysis and (c) a structural response analysis. In addition, these components are mutually coupled by coupling steps. Two approaches to coupling exist. In a one-way coupled approach, data is transferred from the CFD simulation to the FE model, whereas in two-way coupling, the data produced by the FE model is returned to the CFD simulation. The European research project "FIRESTRUC" compared both coupling approaches in predicting thermo-mechanical behaviour. Related to the aforementioned project, Welch *et al.* (2008) present an overview of approaches for coupling CFD and FE models, taking into account accuracy and computational costs. Luo *et al.* (2010) developed a Fire Interface Simulator Toolkit (known as AFIST) by integrating FDS with a customized FE model in Abaqus. A two-way coupling between the fire simulation and heat transfer models has been implemented where heat and mass flow is exchanged at the incremental level. In addition, various demonstration and validation methods are presented to illustrate the capability of AFIST.

With respect to the data transferred from a CFD simulation to a FE model, the concept of the Adiabatic Surface Temperature (AST) was introduced by Wickström *et al.* (2007). The adiabatic surface temperature is a practical approach to express by a single quantity the thermal exposure of a surface to fire, thereby reducing the amount of data flow in a coupled analysis. Duthinh *et al.* (2008) utilized the AST to develop an interface between FDS and FE program Ansys. They then applied this interface to a simulation of a trussed beam under fire and verified it with the corresponding fire test by NIST. Silva *et al.* (2014) developed a Fire-Thermo-mechanical Interface (FTMI), which allows for one-way coupling of an FDS fire simulation to a thermomechanical FE analysis by Ansys. The coupling allows for both convective and radiative heat transfer to the exposed surface of the structure by utilizing AST. Additional validation of the interface was undertaken by Zhang *et al* (2015).

Somewhat different, but worth to mention in this context, is the work of Banerjee *et al.* (2009). They created an Immersive Visualization Environment (IVE) to visualize and study in real time the structural and thermal behaviour of a selected structural element in a one-way coupled fire to thermo-mechanical analysis using FDS and Abaqus.

The above overview illustrates that coupling approaches in literature focus on fire-to-thermal and thermal-to-structural couplings, and in each case both one-way and two-way coupling. But the effect of changes at the building structural level to fire propagation and subsequent further structural failure has not been researched. For instance, failure of a window or a local structural element could result in openings that change the fire behaviour, and consequently influence the fire load on the remaining structural elements.

The contribution of this paper is to show the feasibility of an automated two-way CFD fire-FE thermo-mechanical coupling for global modelling of building structures under fire. In addition, the paper uses the acquired simulation framework to assess the effectiveness of two-way coupling. This was achieved by a study of the failure progression of a thin-walled steel façade using a two-way coupled analysis compared to the same façade using a sequential one-way coupled analysis. In the next section, first the general approach is presented, after which the problem as used for the feasibility and effectiveness study will be given in Section 3. The CFD and FE method are briefly introduced in Sections 4 and 5, respectively, while Section 6 explains the actual programs and scripts. Finally, Section 8 discusses the results and presents conclusions and recommendations.

2. Approach and assumptions

In this paper, one-way coupling is defined as a coupling C1 that transfers data from a Fire Simulation A1 to a Heat Transfer Analysis A2 together with a coupling C2 that transfers data from the Heat Transfer Analysis A2 to a Structural Response Analysis A3. The coupling and processes are illustrated in Figure 1. Additionally, two-way coupling is defined as the couplings C1 and C2 combined with coupling C3, which transfers data from the Structural Response Analysis A3 to the Fire Simulation (A1). Note that it is also possible to refer to two-way coupling, for instance, if coupling C1 is complemented with an additional coupling in the other direction (thus transferring data from A2 to A1), but these types of approaches will not be discussed here.

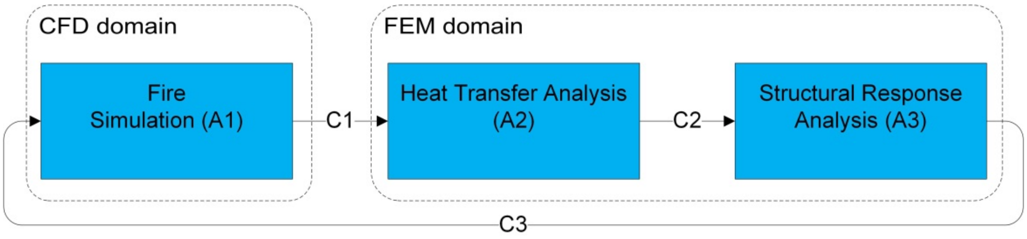


Figure 1: One-way coupling (C1 and C2) and two-way coupling (C1, C2, and C3).

One-way and two-way coupling are fundamentally different. For one-way coupling, a single fire simulation can be carried out for the complete time interval of interest, followed by a single heat transfer analysis along the same interval, and likewise for the structural response analysis. However, in two-way coupling, changes in the building structure influence the fire simulation. A practical approach to take this into account is then to use an iterative procedure, following A1 to A2 to A3 for such small time intervals that changes in the building structure at a certain time "reach" the fire simulation fast enough to ensure a correct simulation.

For the fire simulation "Fire Dynamic Simulator" (FDS) and its accompanying visualization tool "SmokeView" will be used. Hence the time and spatial varying adiabatic surface temperatures will be obtained for later use in the heat transfer analysis. Coupling C1 processes the specific output format FDS such that it can be used as input to the heat transfer analysis. This heat transfer analysis A2 will be carried out by the finite element package Abaqus, taking into account all three modes of heat transfer: convection, radiation, and conduction. The results of the heat transfer analysis are not fed back to the fire simulation, where the building structure is assumed to be adiabatic, making coupling C1 itself one-way as mentioned earlier. Using the specific element package Abaqus for both the heat transfer and the structural response analyses, coupling C2 takes place in the package internally, for which even dissimilar meshes are allowed. Again, this coupling C2 itself is one-way, e.g. heat generation due to deformations and the disturbance of the conductance flow field due to fracture are neglected. Specifically, the nodal temperatures from heat transfer analysis will be applied as boundary conditions to the structural response model. Here, due to temperature increase, the steel elements may expand and buckle. However, the expansion may be partly restricted, thereby generating stresses that could initiate (partial) failure. For this feasibility study, the temperature dependency of the material properties is neglected. This is unacceptable for the realistic modelling of a structure, but believed to be allowable here for a feasibility study. Finally, coupling C3 (see Figure 1) uses a failure criterion, Von Mises, to determine which elements of the building structure fail. These failed elements are then removed in the next run of the fire simulator, heat transfer analysis, and structural response analysis. Due to the discretization size of the fire simulator, it will not be possible to model and study the effect of relative small deformations like those from expansion and buckling.

Given the multiple simulations and couplings, a two-way approach quickly becomes a tedious, time-consuming task. Therefore, C++ programs and a Python script have been developed to both facilitate the coupling steps and manage the complete automatic approach.

3. Experimental setup: model room

The feasibility study involves developing the couplings above and managing the processes A1 to A3. For demonstration and the assessment of the effectiveness of two-way coupling, a model room will be used as shown in Figure 2. A standard office room, normally part of a larger office building, is modelled here as a stand-alone room with an area equal to 5400×3600 mm and a height of 2700 mm. The walls and bottom and top floor are modelled as concrete and a door opening with a height of 2100 mm towards a fictitious corridor is available for ventilation. A widely used cladding type is sandwich panels, built by sandwiching an insulation core between two face sheets, and supported by a steel frame of horizontal beams and vertical struts, as shown in Figure 2. However, detailed modelling of such a thin-walled steel façade system is outside the scope of this research. Therefore, the cladding modelled consists of 4 × 3 monolith steel plates, each simply supported at their horizontal edges as shown in Figure 2 (top right). Failure and removal of a steel plate will result in a direct connection to the outside and thereby will influence the fire.

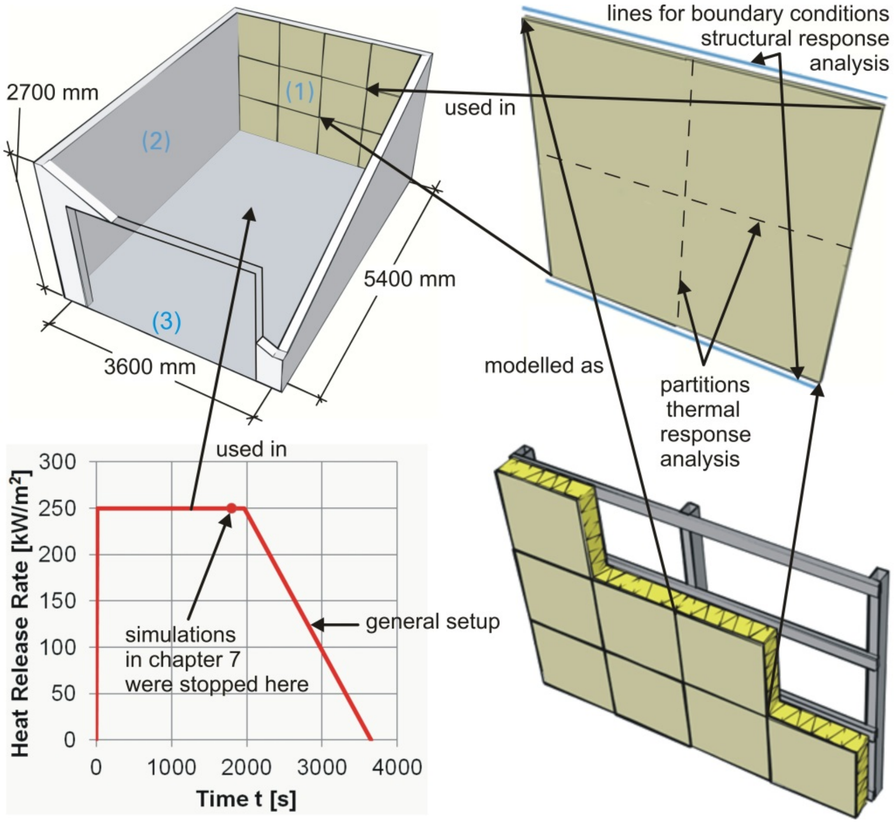


Figure 2: Model Details: Model room (top left) uses steel plates (top right) modelled from a sandwich panel system (bottom right). Room fire is based on graph bottom left.

4. Fire Simulation (A1) with FDS

Fire Dynamics Simulator (FDS) is a computational fluid dynamics (CFD) program that describes the propagation of fire by numerically modelling the fire-driven fluid flow. FDS solves a Large Eddy Simulation (LES) form of the Navier-Stokes equation with an emphasis on smoke and heat transport (McGrattan *et al.*, 2013).

For input a plain text file is used that allows the definition of, among others: 3D rectangular meshes that are meshed with CFD elements (via "&MESH"); 3D rectangular obstructions in such meshes (via "&OBST"); materials via "&MATL", which further specifies surfaces (defined by "&SURF") that are part of meshes or obstructions and holes (by "&HOLE") and ventilation openings (by "&VENT"), which can be applied to surfaces. Several possibilities exist to model a fire, and in this paper a cellulose fire is modelled by defining the heat release rate over time and the enthalpy of formation of cellulose (by "&SPEC").

By default, FDS outputs the total heat release and related quantities to an output file. However, to effectively couple the fire simulation to a subsequent heat transfer analysis, adiabatic surface temperature (AST) at specific locations need to be known. For this, at specific positions the so-called devices (via &DEVC") that record a certain quantity, in this case the AST, can be created.

As seen from Figure 1, for two-way coupling using coupling C3 the FDS simulation needs to be paused, modified, and restarted several times. To stop FDS a dummy file can be generated in the file system, or a device (via &DEVC) can be defined so that a so-called control function (via "&CTRL") is triggered at a certain time that halts the simulation. Modification needs special attention since the FDS user manual states that between stops and restarts, changes are limited to those attributes that do not directly influence the existing flow field. Of course, this is a rationale for CFD simulations in general, and possibly therefore FDS does not allow the removal (or addition) of obstructions during a simulation. This limitation can be solved with the following technique although the resulting simulations should be checked for undesirable effects of disturbing the flow field: First of all, similar to the above device that triggers a control function for a stop, such a device can also be applied to (de)activate an obstruction after a certain time. With this knowledge, at the start of the two-way coupled simulation such a device could be defined for all structural elements that possibly have to be removed, and such that the time the device will trigger will be past the intended total simulation time. If during the simulation the structural response analysis will find a certain structural element to fail, for the fire simulation the time the device associated with this element will trigger will be set to a time directly after the restart. This implies that the fire simulator's time step size is a measure for the delay of removing a failed structural element, and thus should be controlled.

5. Heat Transfer (A2) and Structural Analyses (A3) with Abaqus

For the heat transfer analysis, each panel of the sandwich system is split using four Abaqus "partitions", followed by the formulation of four surfaces, as shown in Figure 2, as this specifically for Abaqus allows for four independent temperature loads along the surface of each panel. For meshing, eight-node shell elements are used, numerically integrated 3×3, and named "DS8" in

Abaqus. The temperature load is applied by using the so-called "interaction module" of Abaqus. In this module, for convection a surface film condition is defined. This condition uses a convective heat transfer coefficient and the sink temperature over time is defined by the adiabatic surface temperature (AST) from the fire simulation. For radiation to the environment, the surface radiation is defined by the emissivity and by the AST defining the ambient temperature over time. Using the adiabatic surface temperatures, it can be shown that the correct total heat flux as a result from the fire simulation is used for the heat transfer analysis.

The structural response simulation models each panel of the sandwich system as a thin steel plate meshed with eight-node shell elements, numerically integrated 2×2, in Abaqus known as the "S8R" element. Likewise as shown in Figure 2, every panel is simply supported along the bottom and top vertical lines, simulating the frame beams. Since the steel plates may buckle due to the thermal stresses, an initial imperfection shape is applied using a scaled first buckling mode before the simulations start. With respect to the thermal loading, nodal temperatures in time from the heat transfer analysis are directly used in the structural response simulation, without any conversion needed.

So far, due to the specific pre-processor used, Abaqus CAE, the steel plates in both the heat transfer and structural response simulations are fully independent. This could be the case practically (Figure 2), but it should be possible to model thermally and structurally coupled plates too. Specifically for Abaqus, this can be carried out by applying the so-called tyings between the surfaces defining the plates, or only applicable to the structural response simulation, by using tyings between the edges defining the plates. However, as will be explained in Section 7, it has been difficult to implement a fully automatic two-way coupled analysis for tied plates.

A restart can be carried out in Abaqus by requesting a restart file to be created regularly and removal of a steel plate is realised by defining an Abaqus "model change interaction" from the interaction module at the beginning of a (restart) step, and selecting the geometry or element region be deactivated.

6. Programs and Scripts

In the previous sections, an approach was discussed for two-way CFD fire - FEM thermo-mechanical coupling for global modelling of building structures under fire. Although inevitably the CFD and FEM programs to be used were mentioned in Sections 4 and 5, the focus was still on the approach itself, potentially applicable to all types of CFD and FEM programs. This section explains how the approach has been implemented, thus including all the specific details of the programs used such that the simulations can be run fully automatically. This latter aspect is important as only a fully automatic simulation allows for correct and extensive parameter studies and comparisons.

Figure 3 shows the implementation of the automated two-way CFD fire - FEM thermo-mechanical coupling. In the centre is the console program FDS-2-Abaqus, written in C++. User input involves the number of plates, number of partitions, simulation duration, iteration step size, failure stress, failure number of integration points, and failure number of finite elements (explained below). With this input, the program runs iteratively the fire simulation, the heat transfer, and the structural response analyses together with the coupling programs as shown in Figure 3. As such, multiple one-way coupled CFD-FEM analyses are carried out. As the CFD and FEM simulations have been discussed in previous sections, the five coupling programs will be presented briefly in this section. An in-depth discussion can be found in Feenstra (2016).

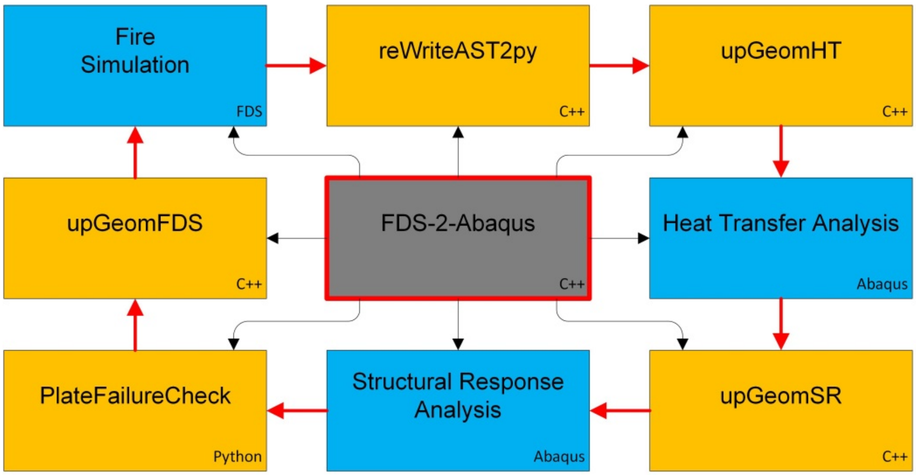


Figure 3: Implementation of fully automated two-way CFD fire-FEM thermo-mechanical coupling.

After the fire simulation, FDS outputs a comma separated values (csv) file that is *time ordered*, i.e. it lists for each time value in the first column, the adiabatic surface temperature for each device in subsequent columns (in the heat transfer analysis, each plate is divided by partitions in four equal surfaces, and at the location of the middle of each surface, a device is placed in FDS). A device has a unique name "AST_*x*-*y*" with *x* being the plate identifier and *y* the plate partition identifier. The C++ program "reWriteAST2py", run by FDS-2-Abaqus, reads in this comma separated values file and generates a Python script for Abaqus, which is *device ordered*, i.e. it contains for each device location a line with a single list of all time vs. temperature values for that specific device location.

The heat transfer analysis that follows needs input via a Python script and it is this script that is written by the C++ program "upGeomHT". The program starts with a basic heat transfer model that has been written manually before the simulations. In this basic model, a plate has been generated by an Abaqus part, this part has been divided using Abaqus partitions, and the four occurring part partitions have been defined by Abaqus surfaces. Then an instance of this part has been copied along the width and height to generate a wall of plates and each part instance (plate) has been renamed with a number sequentially. Then using the device ordered Python script with the adiabatic surface temperatures, this basic heat transfer model is updated for two aspects. First, for each surface of each part instance, the corresponding time vs. adiabatic surface temperature from the file is used for the surface film condition and the radiation to the environment (see section 4). Second, if plates have failed in the previous iteration, these plates are deactivated via the Abaqus command "ModelChange".

Similar to the above program, the C++ program "upGeomSR" starts with a basic structural response model that has been written manually before the simulations. In this basic model, a plate has been generated by an Abaqus part too, but without further partitioning. Then an instance of this part has been copied along the width and height to generate a wall of plates, and each part instance (plate) has been renamed with a number sequentially. Only if a first iteration takes place, an imperfection that follows the first eigen mode is applied. The program adds to this basic model a line that formulates a temperature load imported from the heat transfer simulation "odb"output file. Also, if plates have failed in the previous iteration, these plates are deactivated via the Abaqus command "ModelChange".

After the structural response analysis, the Phyton script "PlateFailureCheck" reads in the related Abaqus "odb" output file. For each time step, the Von Mises stress is read at each FE integration point for which output data was requested (here only the points on the outer shell surfaces). If a certain number of its integration points are yielding, the finite element is regarded as failed. Likewise, if a certain number of finite elements of the plate has failed, the plate itself is regarded as failed.

Finally, the C++ program "upGeomFDS" processes the outcome of failed plates and generates an input file for the next run of the fire simulator. This input file is based on a basic input file that should have been written manually before the simulations. It contains the geometry and building elements of the room, the obstructions to allow for obstruction removal (the plates), and the devices to output the adiabatic surface temperature at the middle of the plate partitions. The file is amended by the program with devices that define a "setpoint" just after the restart for the obstruction that represents a plate that has failed. This results in the removal of this obstruction quickly after the restart of the fire simulation.

7. Results and Discussion

The implementation as described in Section 6 has been tested for the model room shown in Section 3 and Figure 2. First, all relevant variables values used for the simulation will be listed, after which most important test results will be presented.

For the fire simulator, the 5400×3600×2700 mm model room was meshed within a 9000×3600×2700 mm mesh consisting of 30×12×9 CFD-elements of 30×30×30 mm. This left 1800 mm at each side of the length for modelling the corridor and outside environment, separated by a concrete wall with a door opening 2400 mm wide × 2100 mm high, and the steel cladding, respectively. A fuel controlled cellulose fire was set up, working at the complete ground area of the room, with formation enthalpy -5.13E2 kJ/mol. The heat release rate was immediately 250 kW/m^2 at 10 s and continued to 1970 s, then linearly decreased to 0 at 3650 s. Concrete walls, floors and ceiling were used, with a thickness equal to 300 mm, density 1800 kg/m^3, conductivity 1.15 W/(mK), specific heat 1.00 kJ/(kgK), and emissivity 0.8. A corridor wall was defined by a concrete obstacle including a hole for the door, and the cladding was modelled by using for each plate an adiabatic obstacle including a device to be able to remove the plate. Finally, at four locations of each plate, devices were defined to monitor the adiabatic surface temperature.

The heat transfer analysis modelled the 12 plates untied, i.e. independently with no tyings between them, the reason to be discussed at the end of this section. Every plate was modelled by 6×6 shell elements DS8, 0.003 m thick, with 8 nodes and 9 integration points. The material used for these plates was steel, with a Young's Modulus equal to 2.1E11 N/m^2, Poisson's constant 0.29, density 7850 kg/m^3, the specific heat 452 J/(kgK), the conductivity 53.3 W/(mK), and the expansion coefficient 12E-6 m/(mK). Every plate was divided into 4 temperature partitions of which each with an over the partition constant surface film condition and radiation to the environment, based on the adiabatic surface temperature, as explained in Section 4. An implicit transient heat transfer simulation was carried out, with a time period of 150 s, the maximum number of increments 1000, the initial increment 0.15 s, and the minimum and maximum increments being 1E-3 and 10.0 s. Finally, the maximum temperature change to be allowed in an increment was set to 50.

The structural response took geometrical non-linearity and elastic-plastic behaviour of the steel plates into account. Each plate was modelled by 6x6 S8R shell elements with 8 nodes and 4 integration points. Only for the first iteration, and before the structural response analysis, a buckling analysis was carried out on the system with fully tied plates and a uniform temperature increase. The maximum displacement of the first eigenmode was scaled to the shell thickness (0.003 m) and the resulting deformed geometry was imposed on the untied plates for the structural response analysis. The reason tied plates were used for the buckling analysis was because untied plates resulted in many eigenmodes with identical eigenvalues and therefore inconsistencies in the initial imperfection. The implicit dynamic simulation was carried out with an initial increment 0.3 s, and the minimum and maximum increments being 1E-9 and

25.0 s. Steel plasticity was modelled by a yield strength equal to 3.200E8 N/m^2, followed by hardening given the stress vs. strain points 3.570E8 for 0.002, 3.661 for 0.0157, and 5.416E8 for 0.1351.

The managing program FDS-2-Abaqus was used to conduct five identical simulations using the one-way coupled approach for 1800 s and five identical simulations for the two-way coupled approach. For the latter, 12 iterations were taken of each 150 s. After each iteration of 150 s, the C++ program "PlateFailureCheck" calculated which plates failed during this iteration, and the Python script "upGeomFDS" then removed these plates from the fire simulation for the next iteration of 150 s. A plate was set to fail if 13 of its finite elements failed. A finite element was regarded as failed if the Von Mises stress exceeded the yield stress in at least three out of the eight integration points for which output was requested (on the outer shell element surfaces). Table 1 presents the time at which the plates failed. If no time is shown, the plate lasted until the end of the simulation. Figure 4 presents the data graphically.

Table 1: Time to failure of plates in seconds (For two-way coupling, plate 4 avoids subsequent failure of others, unlike for one-way coupling).

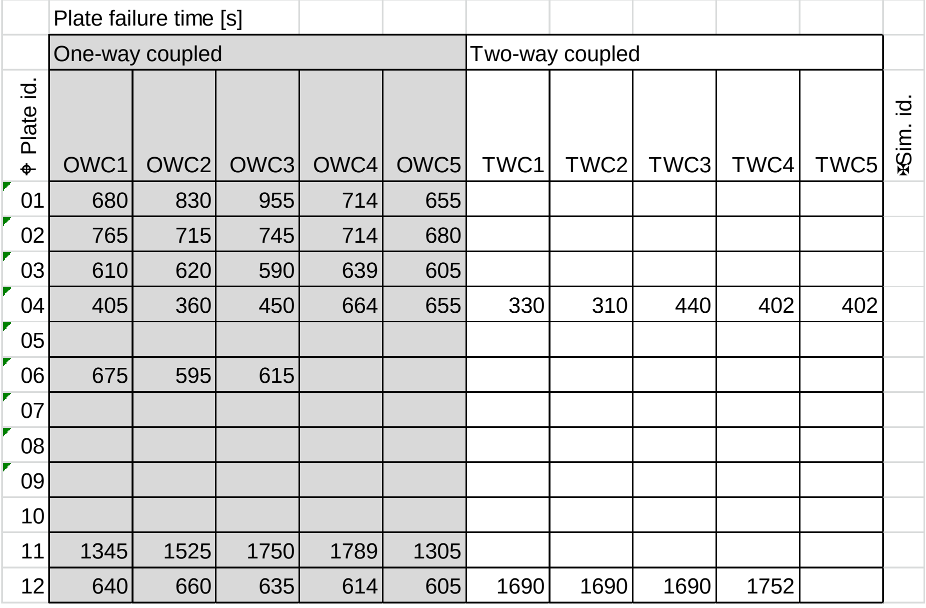


	Plate failure time [s]										
	One-way coupled					Two-way coupled					
Plate id.	OWC1	OWC2	OWC3	OWC4	OWC5	TWC1	TWC2	TWC3	TWC4	TWC5	Sim. id.
01	680	830	955	714	655						
02	765	715	745	714	680						
03	610	620	590	639	605						
04	405	360	450	664	655	330	310	440	402	402	
05											
06	675	595	615								
07											
08											
09											
10											
11	1345	1525	1750	1789	1305						
12	640	660	635	614	605	1690	1690	1690	1752		

Two main conclusions can be drawn from the results. First of all, differences in results exist between exactly the same simulations, having exactly the same input-files and run on the same computer. This is thought to be caused by some randomness in the FDS simulations. However, for the problem investigated here, the qualitative behaviour, i.e. plate failure sequences and fire behaviour are the same for all simulations of a certain approach, as can be seen in Figure 4. This allows for the second conclusion, which is that a significant difference in behaviour occurs for one-way and two-way coupled approaches. From a physical point of view, this is straightforward for the specific problem here, however, it stresses the need to further develop simulation approaches in this direction. This is because it may not always be that clear whether the specific problem at hand does or does not need two-way coupling to be described correctly, and experiments, naturally two-way coupled, are often too expensive. Both conclusions have been verified with the same simulations with smaller FDS mesh cell sizes (0.15 x 0.15 x 0.15 m^3).

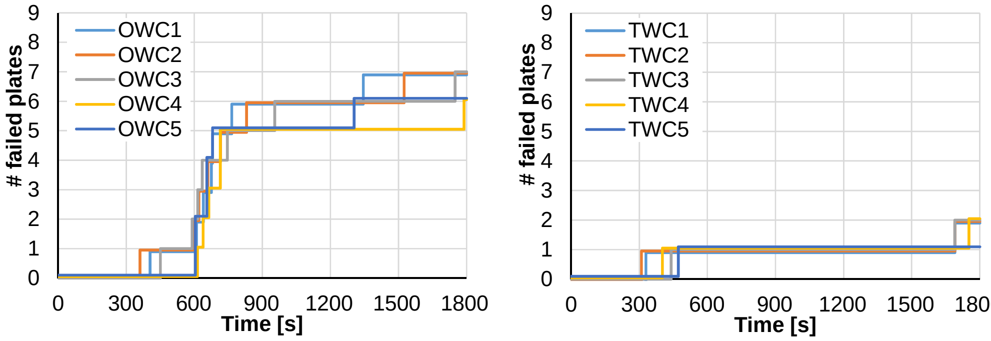
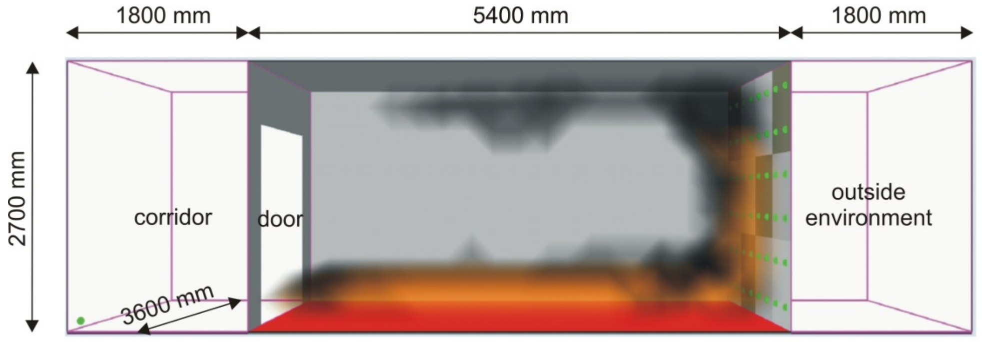


Figure 4: Plots of failed plates versus time in seconds (One-way coupling shows quite a sudden failure of several plates (left), while two-way coupling shows only the second plate failure long after the first (right)).

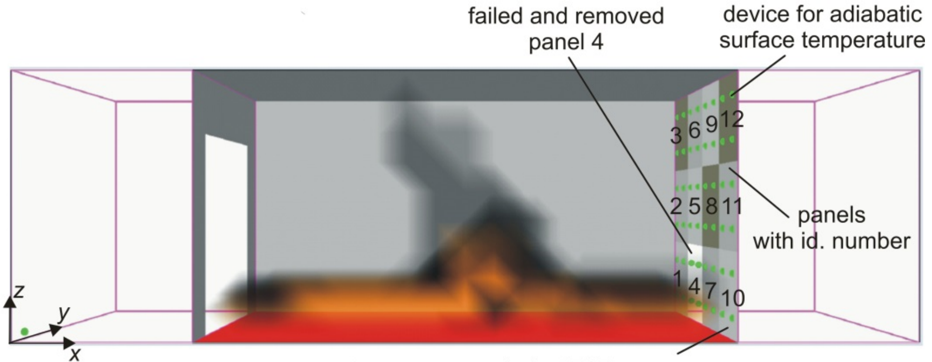
To investigate another fire scenario, the above simulations were carried out again, but with only allowing the eight most upper panels to fail (panels 2,5,8,11 and 3,6,9,12) to avoid the bottom row panels to fail as seen in the above simulations. In the new simulations, the two top row panels in the corners failed around 650 to 800 seconds both for the one-way and two-way approaches. Similar to the above simulations, for the one-way coupled approach two more panels (directly below the previous failed ones) failed later, whereas for the two-way coupled approach all the remaining panels lasted, probably due to hot fumes able to escape. As a conclusion it can be suggested that two-way coupling is not just relevant for a very specific case, but for all cases where a structure collapses (partly).

It was found that slight variations of initial structural imperfections, interval of output requests, and iteration time step size may influence the plate failure times and progression. Therefore, a detailed parametric study, possibly verified using real life fire simulations, should be carried out in the future to further investigate the conclusions and suggestions made here.

Note that in the fire simulations a fuel-controlled fire was modelled. However, real compartment fires are typically ventilation controlled after flashover. This is relevant particularly in these types of two-way coupled analyses, as plate failure in a ventilation-controlled fire would result in additional oxygen, causing backdraft and an overall increase in the heat release rate. These effects could accelerate the failure of the remaining plates. It was found that in order to model a ventilation-controlled fire in FDS, a more complex pyrolysis model than used here needs to be developed (see Section 8.5 in McGrattan *et al.* (2013)).



one-way coupled at 710 s.



two-way coupled at 715 s.

Figure 5: Smokeview visualisations: failed panel # 4 causes the fire to move to the centre thereby reducing the load on the remaining panels.

Finally, with respect to tied plates in the heat transfer and structural response analyses, several attempts have been made to implement these tied plates. Besides the added complexity for all the programs involved (see Figure 3), the main challenge appeared to be the correct removal of a panel. For the heat transfer analysis, plates could be tied using master-slave surface definitions. However, when a plate with a master surface was removed, the plate with the slave surface, now without control, caused zero pivots. In the future, this could possibly be solved by either deleting the associated ties as well (but this needs a complete rebuild of the model, and a simple restart is not possible), or modelling the cladding out of one part (not possible for systems with components), or lowering the conductance such that effectively the panel "vanishes". For the structural response analysis, edges instead of surfaces were to be used for the master-slave definitions. This implied that removing a plate with master edges was without problems, but removing the associated ties caused the underlying boundary conditions to disappear for the plate left, which led to rigid body movements. Also, removing a panel led to immediate large changes in the displacement field and consequently numerical problems.

As such a relaxation step is suggested or, also solving the problems with the ties in lowering of the stiffness such that the plate will effectively vanish.

8. Conclusions and Recommendations

This paper has shown that a fully automated two-way CFD fire - FEM thermo-mechanical coupling can be implemented for global modelling of building structures under fire. To the best knowledge of the authors, this implementation is the first of its kind in allowing for significant changes to the building structure during simulation.

Use of the implementation illustrated a significant difference in the failure progression of a cladding between one-way and two-way coupled analyses. This difference was governed by the change in fire propagation due to geometric updates in the fire, heat transfer, and structural response models. Hence the importance of two-way coupled analysis cannot be neglected, even if it is only for specific cases.

The implementation can be seen as a preliminary framework for the use of two-way coupling in the field of structural fire and safety engineering. It could contribute to a better understanding of both structural response to fire and the response of the fire propagation following these structural changes. Therefore it could become a powerful tool in studying the specific scenarios that would be virtually impossible, or economically impracticable, to investigate using real life in-situ experiments.

Limitations of this research are found in the initial assumptions (e.g. temperature independent material behaviour), the lack of validation, and the specific applications using FDS and Abaqus. However, the overall concept and approach are also applicable to other CFD and FEM codes. In addition, FDS-2-Abaqus and its subprograms are, in its current state, limited to studying structural systems consisting of plates.

Recommendations for future research start with a more detailed fire simulation, for example including ventilation controlled fire, a modelled interior, and a complete fire scenario from the initial phase to the cooling phase. Tied plates for the heat transfer and structural response analyses, including temperature dependent material behaviour, are also needed. The cladding is now over-simplified, which should be modelled with beams and struts, real sandwich panels, and connections, the effect of latter being small, but not unlikely to be responsible for losing a panel. Validation is required, and the implementation could be developed for other types of structures also.

9. References

Banerjee, D., Gross, J., Hess, W., Olano, M., Terrill, J. (2009). *Visualization of structural behavior under fire*: National Institute of Standards and Technology.

Baum, H.R. (2011). Simulating fire effects on complex building structures, *Mechanics Research Communications* 38: Elsevier. 1–11.

Duthinh, D., McGrattan, K., and Khaskia, A. (2008). Recent advances in fire–structure analysis, *Fire Safety Journal* 43: Elsevier. 161-167.

European Committee for Standardization (2012). *EN 1991-1-2, Eurocode 1: Actions on structures – Part 1-2: General actions – Actions on structures exposed to fire*: European Committee for Standardization, Brussels, Belgium.

Feenstra, J.A. (Hofmeyer, H., Mahendran, M., Van Herpen, R.A.P.) (2016): *FDS-2-Abaqus, C++ managed automated python scripted CFD-FEM coupling, Additionally assessing two-way coupling effectiveness*, M.Sc.-thesis Eindhoven University of Technology, Department of the Built Environment, Structural Design Group, The Netherlands, **IFV-VVBA Thesis Award**.

Luo, C., Chen, L., Lua, J., Liu, P. (2010). Abaqus Fire Interface Simulator Toolkit (AFIST) For Coupled Fire and Structural Response Prediction. *51 st AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics, and Material Conference: American Institute of Aeronautics and Astronautics, Orlando, USA*. 1-20.

McGrattan, K., Mcdermott, R., Hostikka, S., Floyd J. (2013) *Fire Dynamics Simulator (Version 6)* User's Guide: National Institute of Standards and Technology, USA.

National Institute of Standards and Technology (2005). *Federal Building and Fire Safety Investigation of the World Trade Center Disaster: Final Report of the National Construction Safety Team on the Collapses of the World Trade Center Towers (NIST NCSTAR 1):* National Institute of Standards and Technology, MD, USA.

Prasad, K., Baum, H.R. (2005). Coupled fire dynamics and thermal response of complex building structures. *Proceedings of the Combustion Institute* 30: Elsevier. 2255-2262.

Silva, J.C.G., Landesmann, A., Ribeiro, F.L.B. (2014). Interface model to fire-thermomechanical performance-based analysis of structures under fire conditions. *Proceedings 2nd Fire and Evacuation Modeling Technical Conference (FEMTC), Gaithersburg, Maryland, USA*: Thunderhead Engineering. 1-9.

Welch, S., Miles, S., Kumar, S., Lemaire, T., Chan, A. (2008). FIRESTRUC - Integrating advanced three-dimensional modelling methodologies for predicting thermo-mechanical behaviour of steel and composite structures subjected to natural fires. *Fire Safety Science* 9: International Association for Fire Safety Science. 1315–1326.

Wickström, U., Duthinh, D., McGrattan, K. (2007). Adiabatic surface temperature for calculating heat transfer to fire exposed structures. *International Interflam Conference, 11th Proceedings, London, UK*. Volume 2: Interscience Communications, 943-953.

Zhang, C., Silva, J.G., Weinschenk, C., Kamikawa, D., Hasemi, Y. (2016). Simulation Methodology for Coupled Fire-Structure Analysis: Modeling Localized Fire Tests on a Steel Column. *Fire Technology* 52: Springer. 239-262.