Sequentially coupled thermal-stress analysis of a new steel-concrete composite slab under fire

Nandor Mago*, Stephen Hicks*, W Ian Simms

New Zealand Heavy Engineering Research Association*, Freelance Consultant (UK)

Abstract: The paper describes the numerical simulations required to estimate the performance of a composite slab using a re-entrant profiled steel sheet, prior to conducting a loaded fire test to verify the performance predicted by the simulations and obtain a fire rating according to accepted international standards. The purpose of the simulations was to optimize the design of the composite slab, by eliminating unnecessary reinforcement bars.

Heat transfer analysis involving convection, conduction and boundary radiation was undertaken on a detailed solid modelled slab as part of a new steel-concrete slab product development. The simulation accounted for the thermal contact resistance at the cold-formed steel-concrete interface. The novelties of the model to incorporate the temperature dependent formulation of the interface thermal conductance provided a realistic prediction of the cross-sectional temperature field, which matched the fire test measurements.

The slab top surface temperatures were assessed according to EN 1363-1:2012 to enable comparisons to be made with the insulation performance criteria. Following the uncoupled heat transfer analysis, the temperatures calculated through the thickness were used as an input into a sequentially coupled thermal-stress analysis of an equivalent shell modelled slab. This was necessary due to the excessive computing time required for the detailed solid modelled concrete slab that was used for the heat transfer analysis. Nevertheless, the calibrated shell that represented the composite slab model accounted for the dissimilar temperature-time (T-t) curves through the slab depth, since the bottom surface, close to the fire source, heats up at a much higher rate than the top surface. From the explicit-quasi static simulation, the load bearing capacity was calculated, which is expressed by the limiting largest deflection and deflection rate for flexural loaded structural members as given in EN 1363-1:2012. From these analyses, it was found that the overall fire resistance was limited by the load bearing capacity criteria.

The predicted insulation and load bearing capacity compared very favourably with the fire test measurements, which are also presented in this paper, and provides confidence in the methodology used in this study.

Keywords: Coupled, Thermal-Stress, Steel, Explicit, Concrete, Heat Transfer.

1. Introduction

The RF55® cold-formed steel product, which is manufactured by M Metal Pte Ltd, is a re-entrant profiled steel deck that is used in steel-concrete composite slabs. The sheeting is an integral part of multi-storey steel-framed buildings in that it performs the following roles: it acts as a safe working platform and protects the workers below; it supports the loads during construction and may eliminate the need for temporary propping; it acts as permanent formwork for the concrete slab; through mechanical or frictional interlock (and/or the provision of end anchorage), composite action can develop with the concrete such that the sheet provides all, or part, of the main tension reinforcement to the slab; and through the provision of through-deck welded stud shear connectors, the composite slab may be used to provide restraint to the steel beams. The RF55® is manufactured from high tensile strength steel (nominal yield strength of 500 MPa) with a base material thickness of 0.75 mm, 1.0 mm or 1.2 mm. The galvanized coating class is Z275 in accordance with AS 1397: 2011.

The composite slab is designed in accordance to EN 1994-1-1 and EN 1993-1-3, while the fire resistance design is typically done in accordance to EN 1994-1-2. Although the finished slab is continuous, in practical design it is normal to consider them to be simply-supported in ambient temperature conditions, so that reinforcing mesh is only provided over the supports for crack control purposes; however, this same reinforcement, together with any supplementary bars, is taken into account when considering the resistance of the slab in fire conditions. Finite element analysis demonstrated that there is no need for two layers of reinforcement to meet the targeted fire resistance rating (expressed in minutes).

The objective of the Abaqus simulations were to estimate the performance of the composite slab in fire conditions, prior to conducting a loaded fire test in order to obtain a fire rating for the RF55® structural decking product according to BS EN 1363-1:2012. The 130 mm deep composite slab with 1.2 mm thick steel decking was required to achieve 120 minutes fire resistance on a prestigious multi-storey building project in Singapore. Figure 1 shows the installation of the RF55® decking on the project together with the slab cross-section.

Figure 1 Installation of RF55® deck in high-rise steel framed building and the cross-section of the steel-concrete composite slab showing the shear studs (right) and mesh reinforcement embedded in the concrete.

2. Sequentially coupled thermal-stress analyses

The existence of gravity/imposed loads and second order effects does not significantly affect the thermal resistance at the steel-concrete interface and, hence, has only a negligible influence in the temperature distribution across the section. When carrying out numerical simulations, this allows a pure transient heat transfer analysis to be first performed to obtain the cross-sectional temperature field, without accounting for effects of the load. Secondly, to apply the calculated temperatures in the time domain to the structural model following the application of the loads, and to obtain its final deformed shape. This is the so-called sequentially coupled thermal-stress analysis instead of a fully coupled analysis, neglecting the possible effect of the displacements over the temperature evolution in fire.

The two-step sequentially coupled thermal-stress analysis was partially performed on a detailed solid modelled profile shown in Figure 2, since the stress analysis required prohibitively long computer time to be performed with the explicit solver (estimated in months); this is to be said in context of a high spec HP Z800 workstation and 10 Abaqus licensing tokens. Therefore, an alternative approach with an equivalent flat shell modelled composite slab was developed, which had its own challenges to perform the simulation realistically. Details of each step of the simulations will be given later.

The specimen floor was of overall nominal size of 4790 mm long by 3005 mm wide. The supporting steel beam centreline distances were 3600 mm and 1000 mm. The furnace apertures are 3000 mm and 4000 mm; thus, one of the edge beams was seated outside the furnace. The floor supported a uniformly distributed load of 5.5 KPa over the 3600 mm span. The continuation section of the floor spanned 1000 mm onto the third steel beam was restrained from vertical lifting (Figure 2). This test set-up has been successfully used in the UK for several years, and is intended to model the worst case of a composite slab at the edge of a building, where continuity is provided over only one support.

Briefly the floor comprised a profiled galvanized steel decking topped with 130 mm thick normal weight concrete. The steel decking material was nominally 1.2 mm thick, grade 500. The 55 mm deep deck was supported on three steel beams and each was attached to the beams with through-deck welded shear studs. The shear studs were 19 mm diameter × 100 mm high (un-welded length), welded one per trough (i.e. at every 200 mm). An 8 mm diameter, 200 mm x 100 mm size mesh reinforcement (B508) was included in the concrete with 25 mm cover from the surface of the concrete. The aggregate for the concrete was limestone. The topping consisted of grade C40 concrete with a nominal compressive cube strength of 40 MPa. It had 2.2% moisture content by mass, which can be taken into account through the specific heat having a peak at 115 °C.

Temperature dependent material properties (thermal conductivity, specific heat, density for concrete, Young’s modulus, thermal expansion, metal plasticity, concrete damaged plasticity,) were assigned to the decking, concrete, shear studs, reinforcement and beams. The relevant standards used were BS EN 1991-1-2, BS EN 1992-1-2, BS EN 1993-1-2 and BS EN 1994-1-2. Certain thermal contact properties are not given in these references. For example, steel to concrete temperature thermal conductance was derived based on the study performed by (Ghojel, 2004) and it is given in Table 1.

Ghojel, 2004 gives a relationship only between the thermal conductance and temperature. Therefore, negligible values were specified for the clearance between the surfaces to make the thermal conductance practically independent from the distance between the closely positioned surfaces.

Similarly, the coefficient of heat transfer by convection was specified on the surface exposed and unexposed to the fire of 25 W/m^2K and 4 W/m^2K, respectively (Franssen, 2010). Thermal emissivity of 0.7 was described on the concrete free faces and steel members, while 0.85 on the 50 mm thick Vermiculate based boards, which protected the supporting beams that consisted of hot rolled I-sections.

Table 1 Thermal conductance as a function of clearance and temperature

Conductance [W/m2K]	Clearance [m]	Temperature [°C]
160.122	0.	20.
123.272	1e-06	100.
108.48	2e-06	200.
103.671	3e-06	300.
101.449	4e-06	400.
100.211	5e-06	500.
99.437	6e-06	600.
98.9151	7e-06	700.
98.5429	8e-06	800.
98.2662	9e-06	900.
98.0537	1e-05	1000.
97.8862	1.1e-05	1100.

2.1 Detailed solid model – transient heat transfer analysis

Firstly, a detailed 3D model of the slab was setup for heat transfer analysis. Shell and solid elements were used to represent the steel decking and concrete slab, respectively. The reinforcing bars and shear studs were meshed by beam elements and were embedded in the concrete. Thermal and mechanical contacts were modelled at the steel-concrete interface in the thermal and stress analysis, respectively. Similarly, contact between the overlapping deck profiles and deck embossments could have been modelled; however, this would have just added further complexity to the analysis. The half model is shown in Figure 2, as the floor sits on the beams supported by the furnace walls.

The radiating 100 mm thick fire block top face was positioned 100 mm below the deck, because in the fire tests the furnace temperature recording through the plated thermocouple defined in EN 1363-1 is also 100 mm below the slab bottom face. The ISO 834 standard temperature-time curve was applied to the fire part.

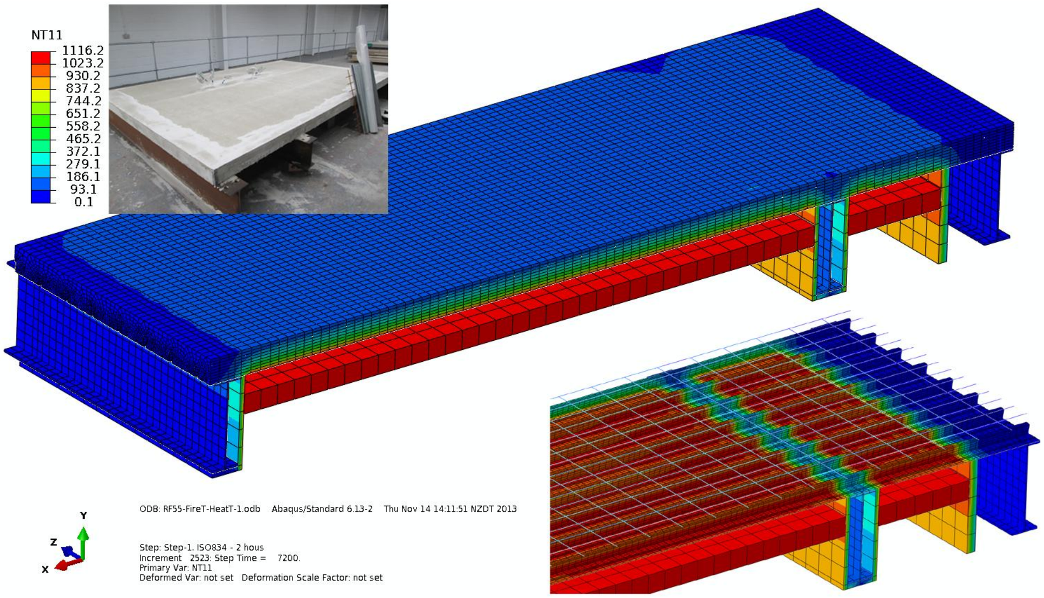

Figure 2 Half of the fire test specimen showing the temperature distribution for 2 hours of ISO 834 fire loading. Photo of the slab waiting to be lifted over the furnace.

2.2 Detailed solid model – stress analysis – concrete cracking

In general, the stress analysis can be performed in implicit or explicit codes.

Often the first choice is Abaqus/Standard's general static, Riks or the dynamic implicit procedure (type=quasi-static). These analysis procedures work fine with the "Concrete Smeared Cracking" and "Concrete Damaged Plasticity" material models unless significant cracking of the structure occurs. As soon as cracks start to develop the solver is forced to take smaller and smaller time increments. It is beneficial to "Specify" "Discontinuous analysis" and to allow more iterations than the default values to take place in the "General Solution Controls Editor" of the Step module. In addition, more tension stiffening and an appropriate non-zero "Viscosity Parameter" in the later mentioned material model allow to push the numerical model further in the material softening and stiffness degradation region. Despite the advanced settings, it might be not possible to obtain a convergent solution. Users should be aware to sensitivity of deflections to tension stiffening. From the present author's experience of analysing reinforced concrete slabs, the deflections are dependent on the amount of tension stiffening, rebar per element ratio (for shells) and other concrete related properties (Mago, 2005). Nevertheless, Abaqus validated implicit and explicit simulations can match experimentally recoded deflection fire time curves and even observed cracking patterns as it is illustrated in Figure 3. The maximum plastic strain vectors (PE) indicate the crack normal vector for the concrete.

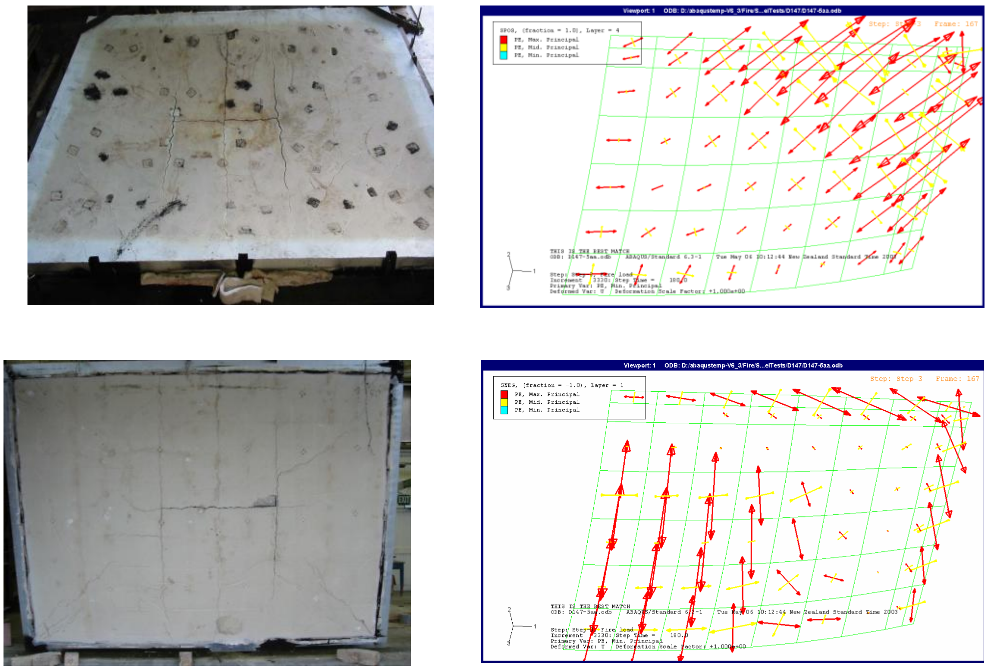

Figure 3 Crack pattern following 180 minutes of fire at the top and bottom face of a flat reinforced concrete slab (D147). The crack plane is perpendicular to the red coloured double arrow. The model is the upper right quarter of the photo (Mago, 2005).

In dealing with concrete, the preferred approach is to use explicit quasi-static procedure, because it is tricky and time consuming to obtain a convergent solution in implicit codes, if it can be calculated at all. On the other hand, the explicit solver might require more time to obtain the solution, but it is almost guaranteed, provided the time increment used for the analysis is sufficiently small.

Therefore, the stress analysis was attempted to be solved with the explicit solver.

The gravity and uniformly distributed pressure was applied in an explicit-quasi static procedure (step-1) using a smooth amplitude curve within 0.4 seconds, based on ten times the period for the eigenmode representing the expected deformed shape. In the subsequent explicit step "Abaqus Answer 3416, Synchronizing the results in a sequentially coupled thermal-stress analysis", was found to be very useful. Namely, in the transient heat transfer analysis the step duration is 7200 seconds. It is impractical to set the time period to such a large time in the stress analysis, because the stable time increment estimate following "Datacheck" is in order of 1e-6 seconds (not mentioning the even smaller time increments were needed). Therefore, based on past experience (Mago, 2008) time scaling of (fire time) up to 1:10,000 proved to be a valuable tool, which means setting the time period to 0.72 seconds in step-2. Regretfully, with even such an accelerated approach, reading in the nodal temperatures from the heat transfer analysis required very small time increments, in some cases in order of 1e-8 seconds or less. The total number of variables in the model was 628641. Attempts to solve the problem with larger time increments (with different mass scaling options) lead to solver deformation speed errors. "Abaqus Answer 1630, Does the load affect the stable time increment in Abaqus/Explicit?" correctly describes a case like this where the external loading (i.e. rapid heating) indirectly influences the stable time increment. Therefore, observing the extremely slow explicit solver (double precision) progress in step-2, an alternative approach has been devised, which is described below.

2.3 Equivalent flat shell modeled slab – transient heat transfer analysis

Since the application of exact temperatures in every node for every time increment of the fire from the detailed heat transfer analysis described in Section 2.1 and 2.2 was prohibitively expensive in terms of computer time, a simplified but conservative approach had to be implemented. An equivalent flat 124 mm thick concrete homogeneous shell modelled slab, which includes the reinforcing mesh and flat decking through Abaqus `*Rebar Layer`, was analysed in a sequentially coupled way.

The flat shell (meshed with DS4 and S4R elements for heat transfer and stress analysis, respectively) was modelled with 13 section points through the slab thickness. Therefore, the corresponding temperatures over time at 13 points were extracted at the slab centre (largest fire exposure) from the detailed solid modelled heat transfer analysis presented in Section 2.1.

It is a good practice to have in mind the number of section points intended to be specified for the equivalent flat shell model, when the detailed solid model is built. Namely, it is convenient to partition the slab at its centre and to assign mesh seeds through the thickness of the slab with an equally spaced number of nodes matching the number of section points in the shell model. This avoids the need for interpolation to extract the Temperature-time (T-t) curves exactly at the shell section points at post-processing stage. This was not noticed early enough in this study, but it was implemented in other similar simulations.

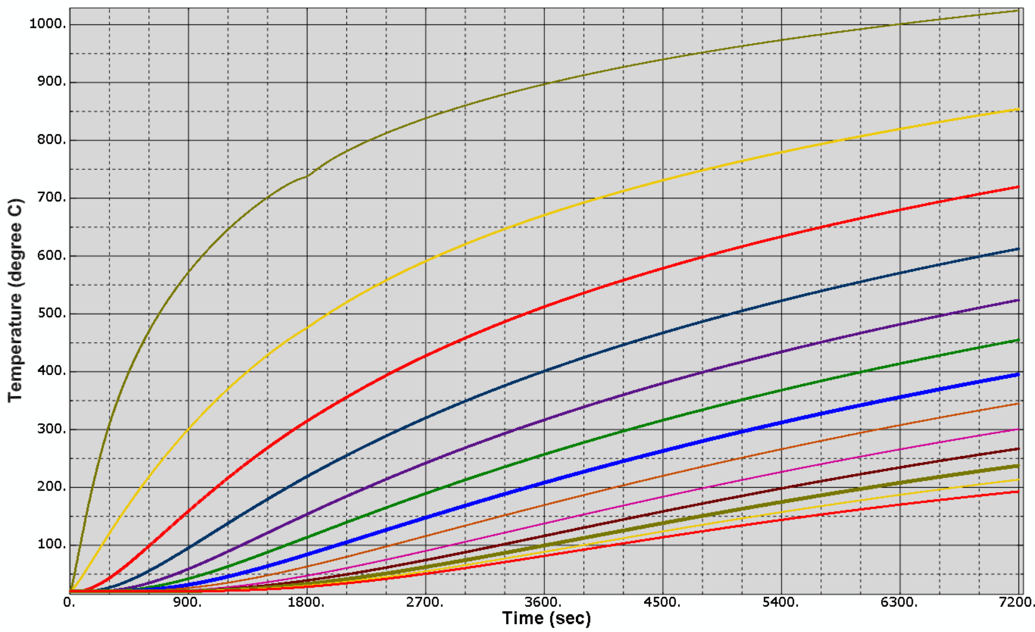

Figure 4 Temperature-time curves through the thickness of the slab at 13 equally spaced points at its centre.

Figure 4 show that heating (and cooling following fire) takes place with different T-t curves through the slab depth. Shell section point 1, closest to the fire, experiences the highest temperatures, while the other curves are lower in magnitude and have different shape. However it is not straightforward to implement this approach in Abaqus/CAE, since it is not possible to simply assign different amplitude curves to different section points of the shell.

The workaround to this limitation is to perform a “fake” heat transfer analysis on the equivalent flat shell model and to read in the temperatures as degrees of freedom into the structural analysis. The amplitude curves shown in Figure 4 were applied as temperature boundary conditions for degrees of freedom 11, 12, 13, ... , 23 to section points 1,2,3, ... , 13, respectively. Following the “fake” transient heat transfer analysis (without any external heat fluxes), the input T-t for each section point are written in the fil file, which is then used in the subsequent stress analysis. It is recommended to add the described approach to the Abaqus documentation.

```
** BOUNDARY CONDITIONS
**
** Name: SlabSecPt1 Type: Temperature
*Boundary, amplitude=SlabSecPt1
SlabExposedToFire, 11, 11, 1.
** Name: SlabSecPt2 Type: Temperature
*Boundary, amplitude=SlabSecPt2
SlabExposedToFire, 12, 12, 1.
…
…
** NODE FILE generates .fil file
*NODE FILE, FREQUENCY=1
NT
*End Step
```

2.4 Equivalent flat shell modeled slab – stress analysis

The simple flat shell model largest deflection due to gravity and imposed pressure load was calculated as 2.5 mm versus 2.6 mm for the solid modelled slab, which was considered to be adequate validation (Figure 5). Being on the conservative side, the slab highest temperatures (as shown in Figure 4) were applied on the whole shell model region above the furnace.

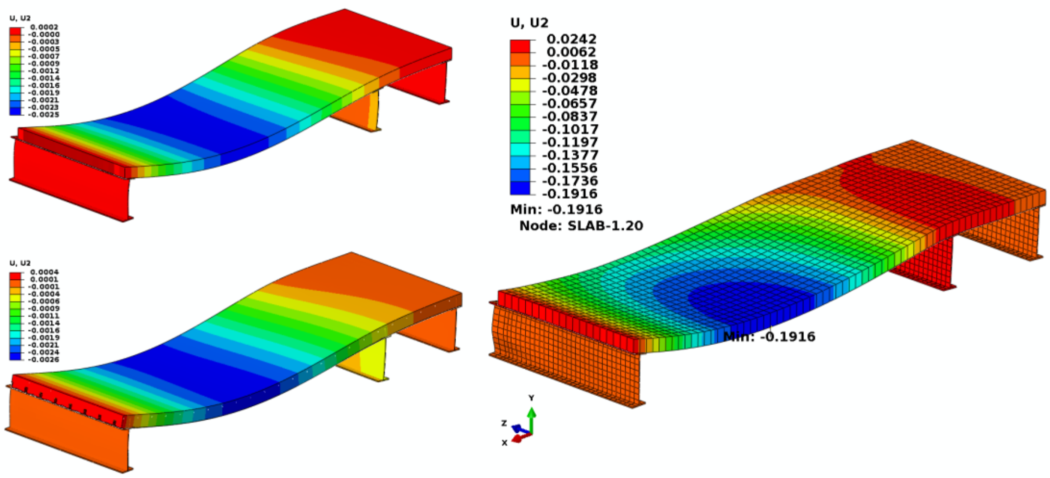

Figure 5 Magnified (100x) sagging (m) contour plot of the shell model (top left) and detailed solid model (bottom left) due to gravity and 5.5 KPa pressure load. Actual deformed shape of the equivalent shell model following two hours of fire (right).

From the analysis based on the 130 mm thick slab, one layer of reinforcing mesh and RF55® 1.2 mm decking profile, based on the requirements of BS EN 1363-1:2012 it is concluded for 120 minutes of ISO 834 fire that:

Performance criteria BS EN 1363-1:2012 – Load bearing capacity:

a) The limiting deflection is 234 mm, based on L=3488 mm, and D=130 mm. The largest expected deflection is 192 mm - PASS

b) The limiting deflection rate is 10.4 mm/min – applicable after the first 10 minutes fire. The largest deflection rate is 3.7 mm/min – PASS.

Performance criteria BS EN 1363-1:2012 - Insulation (Figure 6):

a) The increase in the average temperature above the initial 20 °C is 124.6 °C, which is less than 140 °C – PASS.

b) The increase at any location on the unexposed surface is 169.8 °C, which is less than 180 °C – PASS.

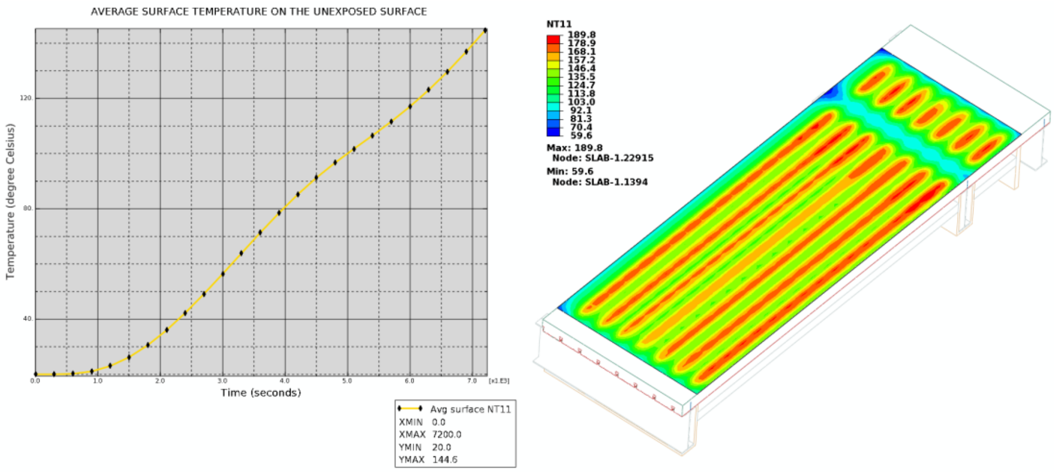

Figure 6 History of average surface temperature (left) and the contour plot of temperature for 2 hours of ISO 834.

It has been observed that mass-scaling (i.e. running the analysis with larger than the stable time increment) could not be used on this problem, due to encountering solver deformation speed check errors. On the other hand, the solver calculated stable time increment had to be scaled down to 10% in the fire step. The scaling factor used under the Abaqus keyword `*Dynamic, Explicit, scale factor=0.1` should not to be confused with the time scaling approach mentioned earlier, which has been successfully used up to 1:10,000. Therefore, the implemented simplified equivalent flat shell modelling approach delivered the results within reasonable time frame.

3. Fire resistance test

The fire tests were performed following the Abaqus simulations. The supporting beams were of smaller size than used in the FEA and the slab measured thickness was 134 mm instead of the nominal 130 mm. Selected photos are given in Figure 7.

Figure 7 The exposed face of the test specimen prior to testing (left) and the unexposed face after 210 minutes of fire.

Temperature recordings at the middle of the 3600 mm span and close to the middle beam at the middle of the 3000 mm span, were taken through the full thickness and above the recess in five minute intervals throughout the 265 minute period of the fire test. The middle beam bottom flange and web temperatures were also recorded. With the availability of actual temperature distribution from thermocouples at two locations, type of aggregate (Calcareous), exact position of the loaded areas, etc, the shell element based analysis was repeated. The tributary area to the thermocouples was assumed. In this instance, the actual slab thicknesses were assigned above the recess and between the recesses. The experimentally recorded temperatures were interpolated and extrapolated to the corresponding shell section points, which is slightly different than what is shown in Figure 4. However, with the knowledge of the in-situ fire test circumstances, the Abaqus simulation predicted a much closer match of the central sagging, which was measured at the middle of the 3.6 m span, as it is shown in Figure 8.

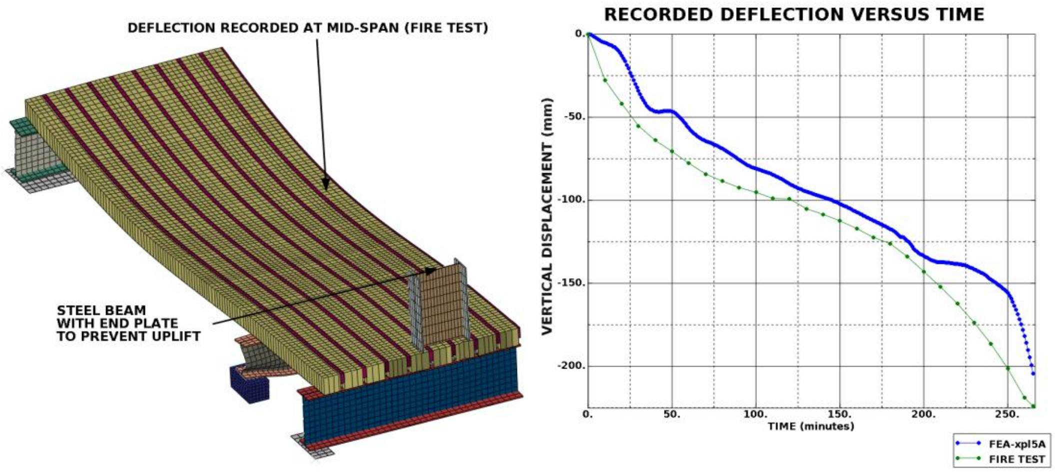

Figure 8 The deformed shape for 265 minutes of fire of the staggered shell model (left), which provided the closest deflection match to the fire test (right).

The in-situ boundary conditions and varying temperature distributions in different regions influence on the numerically predicted deflection-time curve. In addition, the shear studs connecting the floor to the steel beams have most likely failed following 200 minutes of fire as simulations without shear studs showed (deflection time curve not presented herein). Notwithstanding this, the experimentally recorded deflection is of smaller magnitude to that predicted from the simulations prior the actual fire test, therefore making the numerical prediction conservative.

4. Summary

The paper describes the sequentially coupled thermal-stress analysis performed to estimate the steel-concrete composite slab fire resistance rating. The simplified approach, based on an equivalent homogenous shell model but with varying amplitude curves through the thickness, proved to conservatively predict the structural response.

The fire time scaling speed up of up to 1:10,000 has proved to be again a valid approach in performing explicit quasi-static simulations.

5. Acknowledgments

Financial support for this investigation was provided by M Metal Pte Ltd (Singapore). The authors wish to thank the help and advice of Messrs John Kong and Billy Tan of M Metal Pte Ltd, who strongly contributed to the success of the research and development programme. The data used in this paper are the intellectual property of M Metal Pte Ltd.

6. References

1. BS EN 1363-1:2012 Fire resistance tests, Part 1: General Requirements, Brussels, European Committee for Standardization
2. BS 476: Part 20: 1987 Fire tests on building materials and structures, Part 20. Method for determination of the fire resistance of elements of construction (general principles), British Standard Institution
3. BS EN 1991-1-2: 2002 Eurocode 1: Actions on structures – Part 1-2: General actions – Actions on structures exposed to fire, Brussels, European Committee for Standardization
4. Franssen Jean-Marc, Real Vila Paul, Fire Design of Steel Structures, ECCS Eurocode Design Manuals, 2010
5. Ghojel J. Experimental and analytical technique for estimating interface thermal conductance in composite structural elements under simulated fire conditions. Experimental Thermal and Fluid Science 2004; 28:347_54.
6. Mago N. and Clifton G.C., Realistic simulation of reinforced concrete slabs under severe fire attack, Proceedings of NAFEMS World Congress 2005, Glasgow, UK, 2005.
7. Mago N., Clifton G.C., Feeney M. and Porter T., Performance of an Office Building in Fire, Abaqus User's Conference 2008.