Collapse Mechanism of a Space Structure Under Fire Conditions

Shariq Naqvi[1] *Feng Fu* [1,]

Correspondence

Dr. Feng Fu
School of Mathematics, Computer Science & Engineering, Department of Civil Engineering, City University of London, Northampton Square, London, EC1V 0HB; College of Civil Engineering
Email: feng.fu.1@city.ac.uk

Abstract

This paper investigates the collapse mechanism of a long span space structure under fire. The mechanical and thermal properties of a space structure change drastically when subjected to a fire. The nature in which the space structure fails is dependent on a number of factors including fire load, shape and properties of the space structure and its materials. In order to fully analyze how a space structure would fail in these conditions, a multi-physics thermal mechanical analysis model was built in ANSYS software. The chosen prototype space structure, Kings Cross Wester Concourse roof is modeled. The models were created using a series of complex features and were made to be geometrically sound with alignments between each member. Coupled steady state thermal and static structural analysis was performed and collapse mechanism of the space structure under fire were investigated the effectiveness of using finite element analysis software for large assemblies' analysis is also discussed.

Keywords

Space structure, fire, long span structure, collapse mechanism

1 Introduction

It was recorded that around 161,770 fire incidents occurred between April 2016 to March 2017 in the UK alone. There were 261 fire-related fatalities and 7,081 non-fatal casualties in fires. The most recent incident would be the Grenfell Tower fire which occurred 14th June 2017[1]. This breakout fire cause 72 fatalities and over 70 injuries.

When considering safety against the occurrence of a hazard such as fire, the structure being affected by the fire needs to be thoroughly analysed. For a simple structure, one can analyse a simple beam or column within the skeletal structure. However, for a large and complex shape such as a unique space structure, a simple element calculation may not suffice in finding the true nature of its structural response to a certain loading or hazard. In order to investigate structures response in an efficient and accurate manner, the use of finite element analysis is imperative [2-4].

The use of Finite element analysis was primarily in small segments such as beams and elements. Few studies has been perfromed for the gobal behaviour of structure under fire. Fu [4] made a 3D finite element modle to study the global behaviour of a multi-storey building in fire. A study on the WTC 7 (McAllister et al., [5]) made use of ANSYS and LS-DYNA to analysis dynamic structural and pseudo static responses respectively. By considering different temperature and debris-impacts, the several uncertainties around the sequence of events was investigated and the contributors to collapse was able to be identified. This study is one of the most detailed and advanced to date using the numerical approach. However, few studies have been made on the use of finite element analysis for large scale space structures under hazard conditions such as fire. If finite element analysis was to be used for fire conditions, it is often for single elements or small frame. Most of the large assembly analysis are in a 2D plane.

The studies shown in this paper will attempt to contribute to this area of research by finding necessary steps needed to conduct 3D analysis of a large complex structure in hazardous conditions. Therefore, the aim of this paper is to analyse the behaviour a large complex space structure under fire loading conditions using full scaled model in a finite element analysis software.

1. . School of Mathematics, Computer Science & Engineering, Department of Civil Engineering, City University of London, Northampton Square, London, EC1V 0HB, U.K

2 3D finite element model

The Kings Cross Station Western Concourse roof is selected as a prototype structure for the fire investigation [6]. The western concourse structure consists of very complex intertwining geometry. A semi-circular diagrid shell dome structure consisting of rectangular hollow sections and circular hollow sections.

2.1 Geometry set up

The initial method to execute this analysis was to recreate the model in a wireframe. The wireframe of the roof is first developed in AutoCAD as it shown in Figure 1. The supports of the roof are also simulated using columns and bracings.

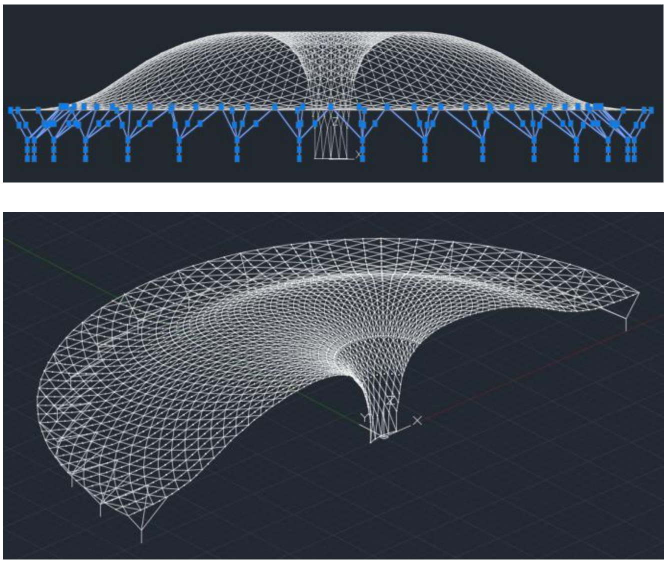
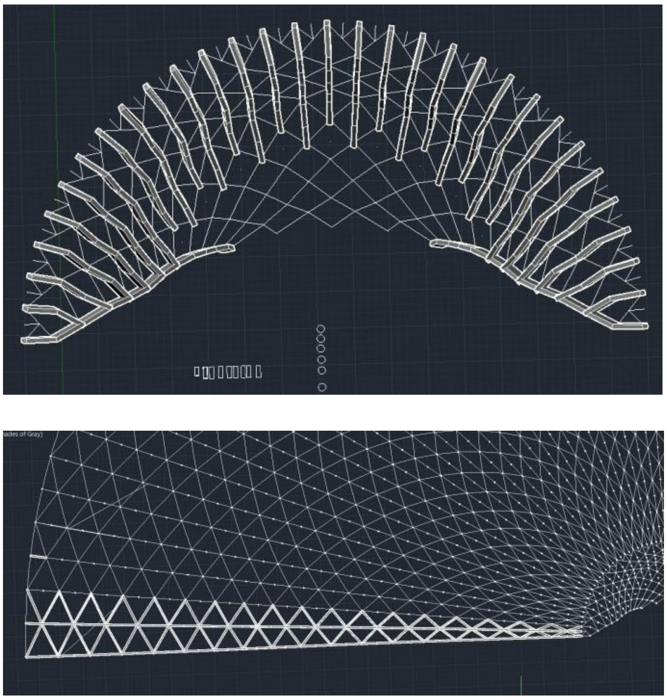


Figure 1 The geometry set up using AutoCAD

During research, a limitation was found that ANSYS, the Finite Element Software, is not able to analyse line bodies using its advanced Multiphysics thermal coupling features. For thermal analysis, apart from temperature increase, it is not able to analyse the nonlinear thermal properties of a material when attached to a line body which are paramount characteristics of a fire simulation. In order to overcome this, the AutoCAD model must be edited, and sections must be added over each line body within the AutoCAD instead of applying sections within ANSYS. The process is shown in Figure 2



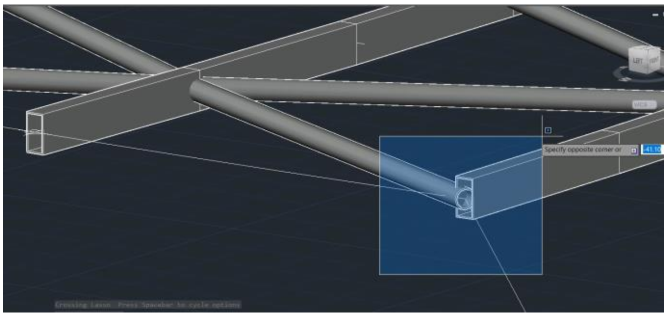


Figure 2 Defined the member section of the roof in AutoCAD

2.2 Finite element model set up in Ansys

After the wireframe of the roof has been set up, it is imported into ANSYS for further analysis. As it is shown in Figure3.

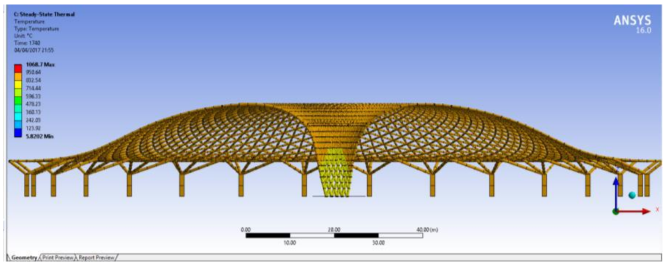


Figure 3 Finite element model in ANSYS

2.3 Time temperature curve

The standard fire Temperature-time curve with various duration was chosen as the fire temperature-time curve. as it is shown in formula 1

$$T = 345 \log_{10}(8t+1) + 20. \quad (1)$$

The above time temperature was worked out using formula (1) in Excel and input into ANSYS. As it is shown in Figure 4.

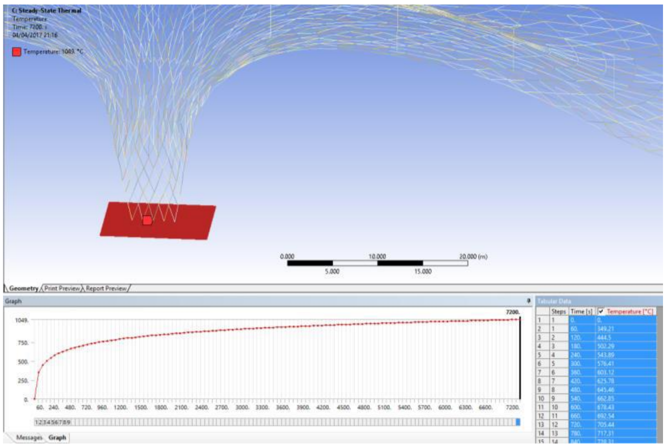


Figure 4 Fire Temperature curve used in analysis

2.4 Material degradation in fire

The material degradation is considered for steel members according to Eurocode. The stress-strain relationship follows Eurocode 3 part 1-2 for steel [7]. The properties of degradation with increasing temperature is shown on Figure 5:

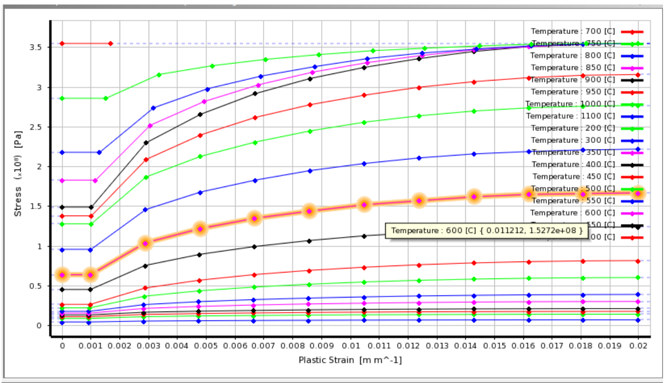


Figure 5 Material degradation of steel under fire

2.5 Heat transfer analysis

To obtain the temperature distribution for the structural models, heat transfer is performed using Ansys Workbench 16 for structural members of the roof. A constant convection film coefficient of 35 W/M^2 and a 0.8 emissivity of radiation were applied to the model.

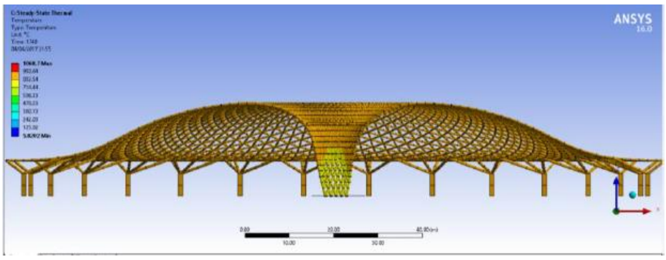


Figure 6 Heat transferring

3 Multiphysics modelling of the roof under fire

The combined Steady-State Thermal- Static Structural analysis can be achieved by the “Coupled Physics Analysis” method available on ANSYS Multiphysics. The first stage is to perform the steady-state thermal analysis, with the inputs of temperature, radiation and convection and time history from the chosen fire scenarios. The body temperature results are then entered into the structural analysis as structural loads. The input data consists of the modulus of elasticity, density, stress-strain relationship and thermal expansion coefficient of steel. This is shown in the figure below.

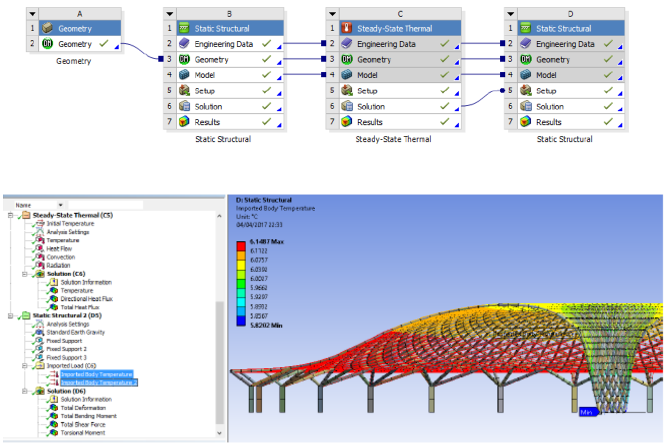


Figure 7 Multiphysics modelling

4 Modelling result discussion

Figure 8 shows the total deformation of each member of the model at a time of 2400 seconds. A graph (Figure 9) have been generated which include the maximum and minimum deflection of the structure (m) at a single time (s). At the time 7200 s the model as a maximum deflection of 1.994 m. the severe deformation indicating the collapse of the dome. The deflection shape can be seen on the model, as the model is highlighted a specific colour with the corresponding value displayed on the side key. It can be seen that the large deformations are concentrated in the centre zone of the roof.

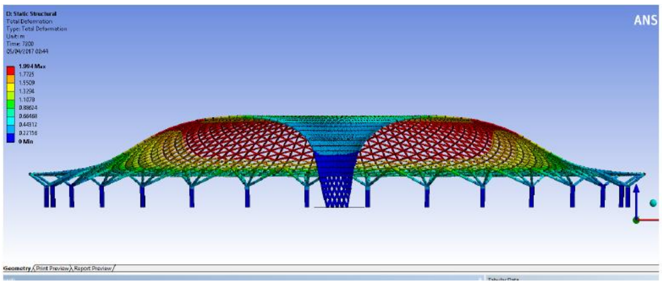


Figure 8 Fire Temperature distribution of the dome

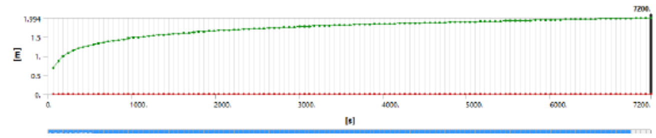


Figure 9 Fire Temperature curve used in analysis

In the past, as it is difficult to set up a 3D model of an entire space structure for investigation its behaviour under fire. Some simplified model such as one planer frame was extracted for investigation the behaviour.

To make a comparative study, a single radial frame is extracted for the analysis. The temperature time curve was able to be mapped to the end of the column faces creating a source of temperature to radiate throughout the model.

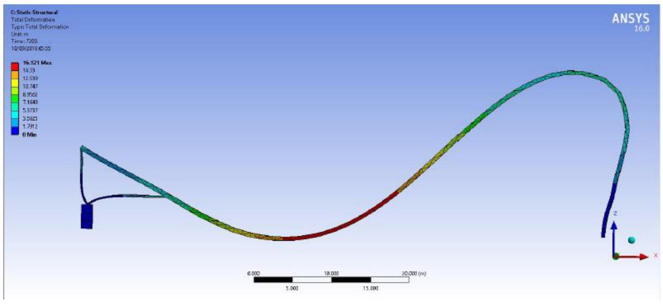


Figure 10 Deform shape of single radial frame

The figure 10 shows the deflection of the model at 7200 second. Due to the large stiffness of the space structure, less deformation was noticed for a space structure under fire than a single frame.

From Figure 8 and Figure 10, It can be seen that, the space structure behaves differently to a single frame under fire load. Therefore, 3D global model needs to be built for clear understanding of the behaviour of a space structure under fire. Most research outcome based on single frame may not be reliable.

Conclusion

From above numerical investigations below conclusions can be made:

1. Multi-physics analysis of the whole structural behaviour provides a more clear and accurate way in investigation the behaviour of a space structure in fire
2. Due to the complex geometry of a space structure, it has dramatic different behaviour compare to a single frame structure in fire condition, therefore, the simplified frame model may not provide accurate results
3. The space structure collapses due to excessive deformation at the center. Therefore, the central region of a space structure needs to be protected under fire condition to delay the collapse of the truss.

References

[1] Fu, F. (2017). How did London tower block fire spread so fast and kill so many? New Scientist.

[2] Fu, F. (2015) Advance modelling techniques in structural design. Chichester: John Wiley & Sons, Ltd. 140-166.

[3] Fu, F. (2016) Structural Analysis and Design to Prevent Disproportionate Collapse.CRC Press. ISBN 978-1-4987-8820-5.

[4] FU, F. (2016).3D FINITE ELEMENT ANALYSIS OF THE WHOLE-BUILDING BEHAVIOR OF TALL BUILDING IN FIRE. ADVANCES IN COMPUTATIONAL DESIGN,1(4), PP. 329–344.

[5] McAllister, T., MacNeill, R., Erbay, O., Sarawit, A., Zarghamee, M., Kirkpatrick, S. and Gross, J. (2012). Analysis of Structural Response of WTC 7 to Fire and Sequential Failures Leading to Collapse. Journal of Structural Engineering, 138(1), pp.109-117.

[6] Shariq Naqvi (2018), Collapse Mechanism of a Space Structure Under Fire and Blast Conditions Master Thesis, City, University of London , Supervisor, Dr Feng Fu

[7] BRITISH STANDARDS INSTITUTION. 2005. BS EN 1993-1-2:2005. EUROCODE 3: DESIGN OF STEEL STRUCTURES – PART 1-2: GENERAL RULES – STRUCTURAL FIRE DESIGN. BRUSSELS. BRITISH STANDARDS INSTITUTION.