International Journal of Infrastructure Research and Management
Vol. 10 (1), June 2022, pp. 1-15

FINITE ELEMENT ANALYSIS ON THE STEEL FIBER-REINFORCED CONCRETE BEAMS: A SYSTEMATIC REVIEW

Hou Zhicheng and Norhaiza Nordin
Infrastructure University Kuala Lumpur, MALAYSIA

ABSTRACT

Steel fiber-reinforced concrete beams have been widely researched, including static mechanical performances, fatigue behaviors, prediction of capability and so on. It is widely accepted that numerical simulations are useful as they reduce more time, materials of casting and testing processes. This paper reviews the effect of finite element analysis on the steel fiber-reinforced concrete beams with commercial software ABAQUS and combines the results of the literature including FEA investigations. More than 150 papers were downloaded and only 8 papers from all the downloaded papers included contents of simulation specimen procedures with ABAQUS. Those papers were seriously compared between each other about what constitutive relations selected and details of defining other parameters in this presented work. What kind of material models in ABAQUS software was used more often and how to define the important parameters in recent study was discussed. Furthermore, this paper discusses of the two most important factors which are simulation accuracy and calculation efficiency about FEA and analyses the influence parameters based on the results of literatures. In brief, FEA simulation of SFRC beams with ABAQUS could be accomplished in good agreement between the experimental and numerical predict results and the discrepancies in general could be less than 10% with suitable data of other researcher's experiment, while the number will be limited to 5% with exact average data of experimental results based on the same group of material. It should be noted that ABAQUS software is valuable and enough accuracy for simulation of SFRC beams and already be general employed in project investigations in recent years based on literatures regarding SFRC beams.

Keywords:
Finite element analysis, steel fiber-reinforced concrete, ABAQUS, accuracy, efficiency, review

INTRODUCTION

Steel fibers can maintain longer useful working life for the elements of reinforced concrete, increase the impact resistance and fatigue endurance. Alongside shrinkage reduction, they also distribute uniformly the stress taking by the SFRC, deduction of surface permeability, dusting and water can also be done. Steel fibers can increase through multi-direction (cross section), while conventional steel rebars only used to improve the concrete strength at single direction. Steel fibers always have been claimed that they are the best crack resistant materials for concrete, since they not only can prolong the crack to be happened and even cracking occurs, but also increase the initial crack strength. Many countries could produce qualified steel fibers around the world, such as HIC Corporation from Korea, Remix Steel Fiber Co. Ltd. from China, KIMMU (Group of Companies) from Malaysia. Steel fibers are needles of wire, deformed and cut to lengths, for reinforcement of mortar, concrete and some composite materials. Some fibers are cold drawn wire fibers with flatted or corrugated shapes. A. Bernard in California (1874) patented the designing of reinforcing concrete with the mean of the addition of steel splinter, since then, the long practice of inventing modern SFRC was began. According to a book (Steel fiber reinforced concrete) wrote by Maidl BR (1918), author patented an approach of modifying SFRC by long steel fibers in France, which could improve tensile strength of concrete. For recent six decades there have been investigated immense research projects devoted to the application of steel fibers and mechanical properties of SFRC. Nowadays, SFRC have become the third important concrete based on structural materials beside concrete reinforced by steel meshes and traditionally reinforced concrete (by stirrups and rods) (Katzer & Domski, 2012). Folino et al. (2020)

ISSN Print: 2811-3608
ISSN Online: 2811-3705
https://iukl.edu.my/rmc/publications/ijirm/
DOI: 10.63646/ijirm.2022100101

analysed the mechanical and failure behaviors of full-scale reinforced concrete beams reinforced with SFRC in their research. As predicted, it was realised that steel fibers contribute to increase the structural integrity in post-peak behaviour, both in structural and small beams. Yoo et al. (2017) have investigated the possibility of excluding the minimum shear reinforcement in reinforced sustainable high-strength concrete beams by addition of 0.75% (by volume) of steel fibers. Their results clearly demonstrated that addition of steel fibers in higher concrete beams would increase the performances of the beams which were identical to conclusions of other researchers (Mertol et al., 2015).
Abaqus company was set up in 1978 by Dr. David et al., since 2014, the headquarters of the company were located in Johnston, Rhode Island, United States. It's important product ABAQUS is a software used for both the simulation and analysis of mechanical assemblies and components (pre-processing) and visualizing the calculation results. Abaqus was originally designed to deal with non-linear physical behaviors, therefore, the package includes a large-scale range of material models such as hyperelastic (soft tissue) and elastomeric (rubberlike) material capabilities. The Abaqus Unified FEA product software offers complete and powerful solutions for both ordinary and sophisticated engineering issues covering an extensive spectrum of industrial applications. First-in-class companies are making use of Abaqus Unified FEA to consolidate their tools and processes, reduce costs and inefficiencies, and obtain a competitive advantage.

LITERATURE REVIEW

Research Objective and Scope of Study

Experimental study of size effects and different parameters of SFRC beams need to so many materials as well as labors since a large number of dimensions of similar elements will be casted to get the mechanical performances. For such objective, the cost-effective numerical simulations always were carried out in recent researches of SFRC beams.

A systematic approach was taken in this paper to investigate the finite element analysis on the steel fiber-reinforced concrete beams. The idea of numerical simulations with ABAQUS was highlightedd in the SFRC beam investigations and industrial products fields in this paper. In the meantime, what kind of material models in ABAQUS software was used more often in recent study was also discussed. Therefore, how to define the important parameters such as the dilation angle, the viscosity parameter and so on in ABAQUS was presented further. Later, simulation accuracy and calculation efficiency, the most concerning factors about FEA with commercial software were compared between different study based on exact data from review papers. Finally, this paper concluded that FEA simulation of SFRC beams with ABAQUS could be accomplished in good agreement between the experimental and numerical predict results.

Review Methodology

This review about the Finite element analysis on the steel fiber-reinforced concrete beams began with the collection and assessment of relevant literatures published in recent decades. For papers collection, famous ScienceDirect and EThOS websites were utilized and updated up to March 2022. The keyword search included "steel fiber reinforced concrete beams" and then title (and abstract) screened. Moreover, other FEA software, for example, ANASYS and DIANA programs were introduced and employed in papers less than that with ABAQUS especially in papers published in recent years. Those papers were seriously compared between each other about what constitutive relations selected and details of defining other parameters for this presented work.

In the progress of comparing, first of all, all papers included in this paper were divided into different groups. The three main categories were: "static behaviors of SFRC beams", "dynamic behaviour of SFRC beams" and "fire resistance of SFRC beams". Every category of papers was simulated to different load of engineering, for an example, "static behaviors" included 3 or 4 point bending test, "dynamic behavior" included low-velocity impact loading test and fatigue behavior test. However, for "fire resistance behavior" category, just one paper (Liu et al. 2018) was found which presented fire load condition of engineering. Most papers discussed in this paper, their authors have not only carried out specimen test, but also calculated the SFRC beams with ABAQUS at the same time. And all their FEA results were very close with their experimental results. Therefore, based on those research of authors, the simulation with ABAQUS software was reliability clearly and strongly. The research process and methodology were shown in Fig.1.

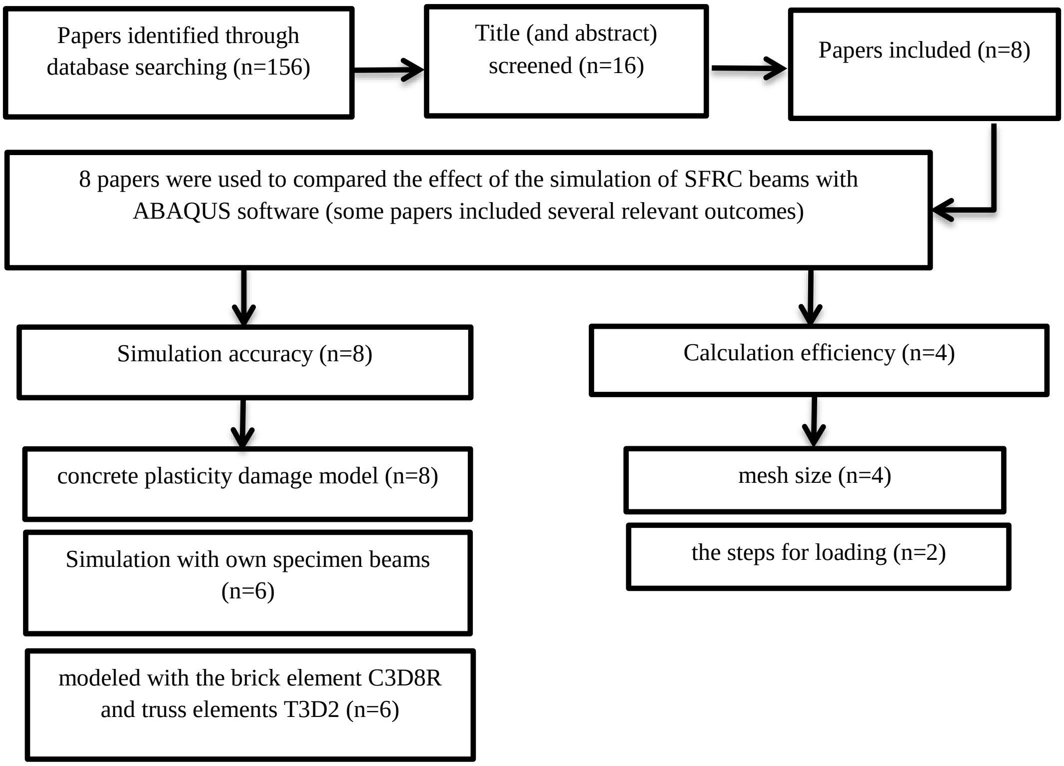


Figure 1: Flow chart of the research process and results.

REVIEW RESULTS

Finite Element Analysis of Static Behaviors of SFRC Beams

Hamoda et al. (2019) investigated numerically and experimentally behaviors of steel-I beam with or without high strength bolted connectors embedded in normal/Steel Fiber-Reinforced Concrete with the finite element package ABAQUS. In their FEA, all constitutive materials including SFRC, NC, steel I-beam, and internal steel reinforcements were created using suitable elements available in the Abaqus software. Furthermore, appropriate contact interactions, meshing properties and boundary conditions based on the performed push-out tests were established. And they employed concrete damage plasticity model to present the inelastic behaviors for both SFRC and NC (see Fig.2).

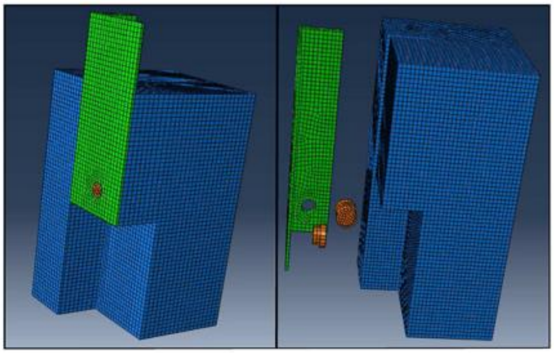


Figure 2: 3D Finite Element Model (Quarter of Specimen) (Hamoda et al. ,2019)

They identified the NC material stress-strain behaviors according to compressive and splitting tests performed experimentally (see Fig.3). The integrity of FEA results was carefully compared and verified against those experimental results with variation about 8% that can be enough valuable for investigating composite section with short demountable boltted connectors (see Fig.4).

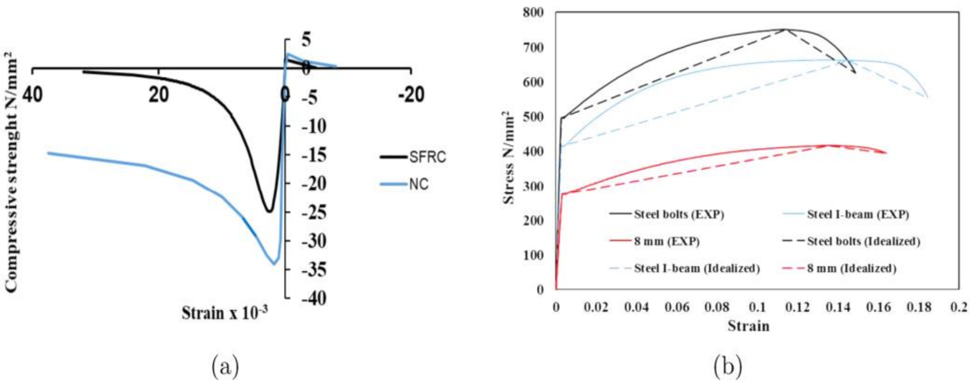


Figure 3: Material Stress-strain Laws Adopted in Finite Element Modeling;(a) Concrete (b) Real and Idealized Uniaxial Stress Strain Relation for Steel Elements (Hamoda et al. ,2019)

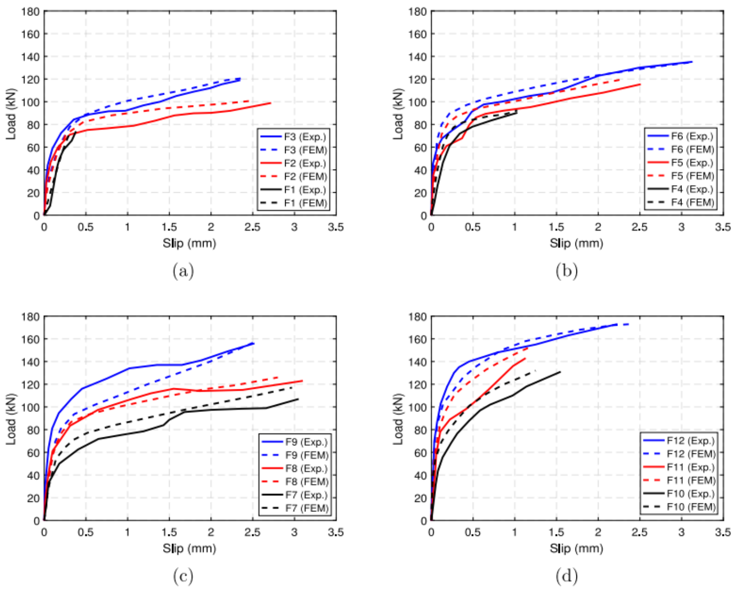


Figure 4: Experimental Versus Finite Element Load-slip Curves for Specimens Made of SFRC (Hamoda et al. ,2019)

Their experiment clearly demonstrated that investigating mechanical performances of some elements with the finite element package ABAQUS is useful and it could be accurately agreement with practical samples. Their process of research in developing nonlinear 3D-FEA model with the finite element package ABAQUS was perfect and could be example in future research about investigation of SFRC beams.

Experimental researches of various SFRC beams were expensive because of a wide range of dimensions of similar beams needed to be casted and tested to gain a full-size effect law. Therefore, researchers often simulated the experimental elements with the versatile, cost-effective numerical models. In research of size effects of Ultra High Performance Steel Fiber Reinforced Concrete (UHPFRC) beams from Mahmud et al. (2013) as in Fig.5, they modeled nonlinear finite element SFRC beams using the concrete damage plasticity model in ABAQUS too. They assumed that steel fibers were evenly distributed in the matrix and the UHPFRC was thus simulated as a homogeneous material.

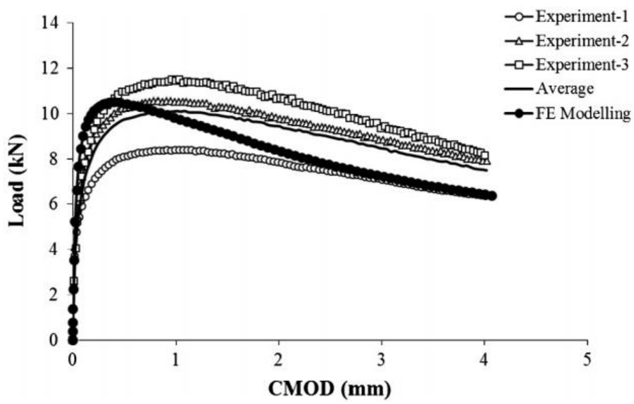


Figure 5: Load-CMOD Curves for Beams with d=60mm (Mahmud et al. ,2013)

They ran the command *CONCRETE TENSION STEFFEINIGN, TYPE=DISPLACEMENT in beams simulation process with ABAQUS for implementing equivalent stress-crack opening displacement (COD) curves. Meanwhile, the compressive strengths of SFRC were modeled with running the command*CONCRETE COMPRESSION HARDENING, TYPE=STRAIN. They noted that the mechanical properties of samples captured from experiments include elastic parts but inelastic quantities were established for the simulation with ABAQUS so transformation was needed. All the beams were modelled with reduced-integration 4-noded plain stress elements (CPS4R).

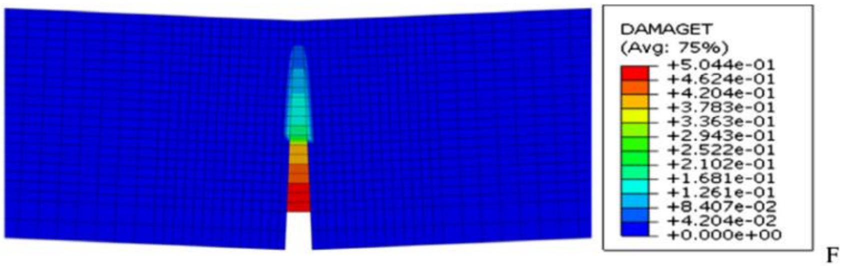


Figure 6: A Deformed Mesh with Damage Contours (Mahmud et al. ,2013)

They compared the three load-CMOD and average curves obtained from their bending tests with the CDP-based numerical results in Fig.4 for beam with d=60mm as an example. Overall good agreements between the experimental and numerical results can be observed. They estimated that the discrepancy maybe attribute to the input pre-load 2 kN which was too close to their peak load of beam about 2.5-3.5 kN. They showed the deformed mesh with damage contours in Fig.6, and their analysis demonstrated that the CDP-based finite element models could predict the bending capacities of the UHPFRC beams with considerable accuracy and therefore parametric studies can further be carried out for size effect analysis (see Fig.7). According to the research of Mahmud et al. (2013), it was proved that a full-size effect law about the UHPFRC beams could be predicted with ABAQUS and pre-load to sample need to be designed carefully in future research.

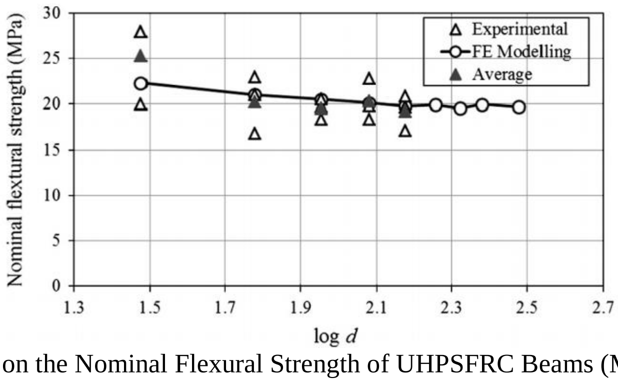


Figure 7: Size Effect on the Nominal Flexural Strength of UHPSFRC Beams (Mahmud et al. ,2013)

Simwanda et al. (2021) have validated nine beams which were conducted by other three researchers by comparing experimental load-deflection capacity and FE predicted load-deflection capacity with ABAQUS software in their study. It was indicated that there was a good agreement in terms of peak load, initial stiffness and corresponding displacement. Shewalul et al. (2021) have modeled a continuous beam with two spans using the FEA software ABAQUS. They calculated the amount of moment redistribution with FEA and verified the results based on experimental tests at

Stellenbosch University. Their FEA models and experimental specimen provided almost similar results with a nearly similar mid span response.

Finite element analysis of dynamic behaviour of SFRC beams

Murthy et al. (2018) created a finite element model to predict the number of cycles to failure and load-deflection behaviour of the RC beams strengthened with ultra-high performance fiber reinforced concrete. The data of compressive stress and strain were quoted from experiments of Mier et al. Further, the constitutive of post cracking relationships for materials were obtained from the works of Almusallam TH et al. They employed the concrete plastic damage model to present the nonlinear behaviour of SFRC. The concrete beam and steel rebars were modeled with elements mentioned earlier. The FE model and typical assembly were shown as Fig.8.

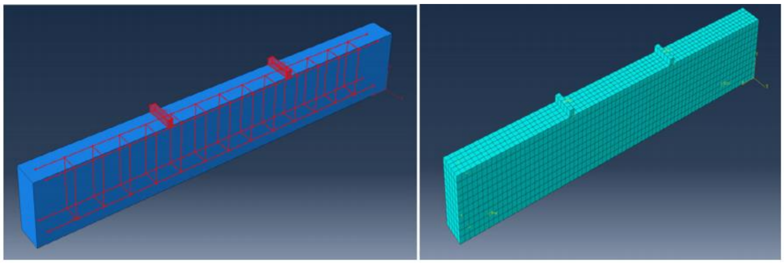


Figure 8: Typical Assembly and FE Model (Murthy et al. ,2018)

Their beams were analyzed with direct cyclic loading with FEA ABAQUS software. The frequency of loading based on the experiments was simulated as periodic amplitude as depicted in Fig.8.

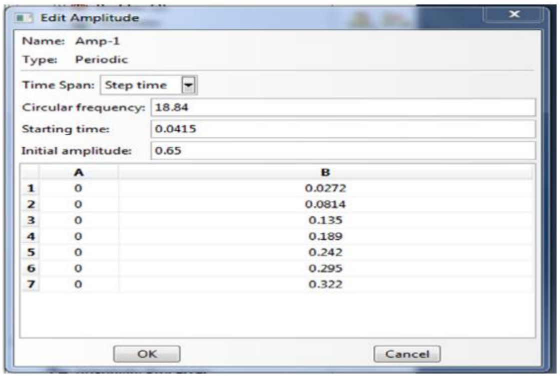


Figure 9: Cyclic Load data input in ABAQUS (Murthy et al. ,2018)

Furthermore, they compared the predicted load-deflection behaviour and the number of cycles to failure with the average experimental results. Fig.9 and Fig.10 clearly demonstrate the parts of results, there was very good agreement between the FEA results and practical observations. Furthermore, the maximum difference from comparison was less than 10%. However, the discrepancy from other researchers was 5% more or less and the more accurately results was because of simulation with data of constitutive relationship obtained from practical experiments carried out by researchers themselves. Therefore, it is widely accepted that simulation SFRC beams in FEA with the data from the same project is best choice and could make the numerical results as closely as possible to the experimental.

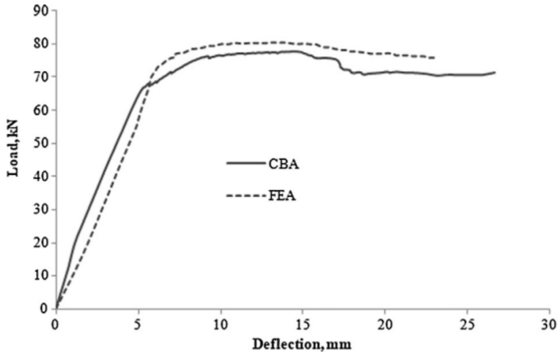


Figure 10: Predicted Load-Deflection Curve of Control RC Beam Compared with Average Experimental Curve (Murthy et al. ,2018)

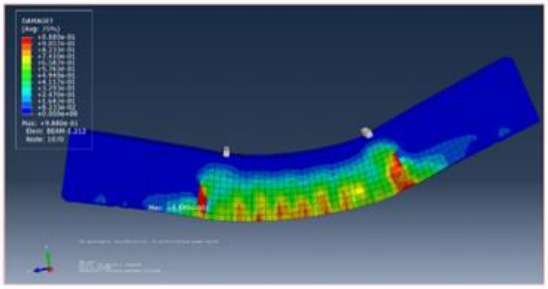


Figure 11: Flexural Damage Pattern for Typical Retrofitted Beam (Murthy et al. ,2018)

Gao et al., (2021) have ever considered fiber clustering in the concrete damaged plasticity model in ABAQUS and simulated the crack patterns of SFRC beam. Their FEA results agreed well with the experimental results, and they suggested that ABAQUS software could be used to investigate the fiber clustering effects in other SFRC structures. Mohsin (2012) had modeled a lot of SFRC beams with ABAQUS software in research of PHD period and calibrated them with experimental results from Campione et al., (2003, 2006) and Mangiavillano (2008). The FEA models yielded the most satisfactory agreement with experimental data.

Finite Element Analysis of Fire Resistance of SFRC Beams

Jin et al. (2018) carried out the experiment of fire resistance of SFRC beams after low-velocity impact loading. In their research, they established a finite element numerical model which considering the effects of high temperature and strain rate. Similar with the aforementioned work, they described the behaviour of SFRC with the concrete plasticity damage model which proposed by Lubliner et al. (1989) and later updated by Lee and Fenves (1998). This model was widely employed for the simulation of static and dynamic behaviors of SFRC. They took into account of the mechanical properties of SFRC which altered by steel fibers and utilized the data from their own experimental tests. About the basic material parameters, such as Young's modulus, strength and strain at peak stress were inputted in model according to Chinese code. It should be noted that the model was meshed by eight-node solid element except that steel rebar was meshed by two-node wire element. At last, the model was meshed to total number of degrees of freedom up to 800000 and the mesh sizes of SFRC and steel rebar were 10mm while other parts of model was meshed to elements of size 30mm which could improve calculation efficiency. The setup of experiment was depicted as Fig.11. The numerical results were presented after simulating two steps which were exactly as same as the practical experiments. The parts of FEA results and comparison conclusions were displayed in Fig.12 and Fig.13.

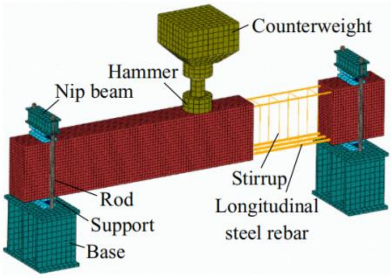


Figure 12: Finite Element Computational Model of the Beam (Jin et al. ,2018)

Based on those results of pictures, it was obviously demonstrated that slight discrepancies were no chance to eliminated totally and the reasons of that already been explained logically by the researchers in their analysis procedures. However, it still can be proved that the FEA method was effective to simulate the main behaviour of SFRC beams exposed to both impact loading and fire.

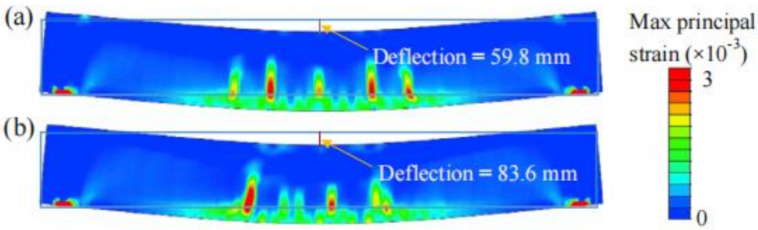


Figure 13: The Effect of Impact-induced-Damage on Fire Deformation of SFRC Beam B-2. (a)Without impact loading; (b) After Impact loading (Jin et al. ,2018)

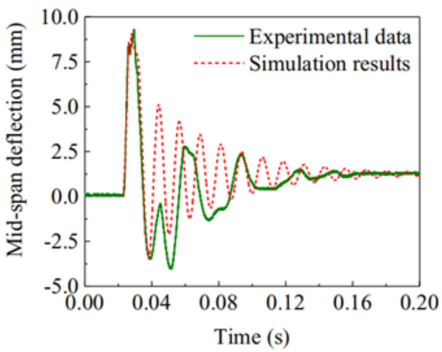


Figure 14: Comparison Between the simulated Mid-span Deflection Time Histories with the Experimental Ones for B-2 (Jin et al. ,2018)

CONSTTITUTIVE MODELS FOR SFRC AND MATERIAL MODELS IN ABAQUS SOFTWARE

Constitutive Models for SFRC

Over the past few decades, many constitutive models for SFRC have been proposed based on practical experiments and theory calculation. To get the post behaviors of SFRC, a number of compressive and flexural samples were tested by researchers. RILEM TC 162-TDF Recommendations (2000) proposed stress-strain relations for SFRC. The compressive strength was subsumed to be improved than that of plain concrete. More recent works for stress-strain relations were done later, RILEM TC 162-TDF (2003) updated the residual flexural strength of SFRC with the following crack mouth opening displacement or midspan deflection values. Moreover, Barros and Figueiras (2001) and Tlemat et al. (2006) had proposed SFRC model separately. Researchers ran their experimental data obtained from cylinders and dog-bone specimens in general as the stress-strain relationship in FEA simulations which were not in the same scales with that of curves they depended (Jin et al.,2018 & Gao et al.,2021).

Material models in ABAQUS software

The concrete damage plasticity model in ABAQUS was used in most study, assuming steel fibers were uniformly distributed in the matrix and then the SFRC was modeled as a kind of homogeneous material (Jin et al., 2018 & Gao et al., 2021). There are about five other parameters needed to be defined in SFRC model basically: in most cases, the dilation angle in degrees is 33, the flow potential eccentricity is 0.1, the ration of initial equibiaxial compressive yield stress to initial uniaxial compressive yield stress is 1.16, the viscosity parameter is 0.66 and the ration of the second stress invariant on the tensile meridian to that on the compressive meridian is 0 (Mahmud et al., 2013). These values were insignificant to the results of simulation based on opinion from Mahmud et al. (2013) and Wang et al (2017). Further, the SFRC or the beams were modeled with the brick element C3D8R (Cube Three Dimensional eight-node Reduced integration) in many investigations and the reinforced steel rebars were generally modeled with three dimensional truss elements (T3D2) which having three translational degrees of freedom at each node (Gao et al., 2021).

RESULTS AND DISCUSSION

Simulation Accuracy

It is obviously that the most concerning factors about FEA with commercial software are simulation accuracy and calculation efficiency. It is no chance to make FEA simulation absolutely equal to practical experimental results as even performances captured from two same beams with total same designation cannot be the same. The slight discrepancy may be from the assumption of material homogeneity, different properties and loadings and so on (Mahmud et al., 2013).

ISSN Print: 2811-3608
ISSN Online: 2811-3705
https://iukl.edu.my/rmc/publications/ijirm/

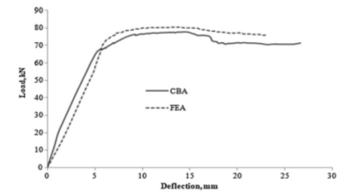
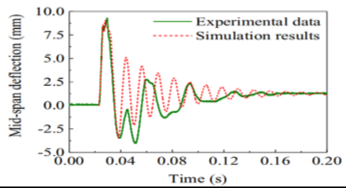
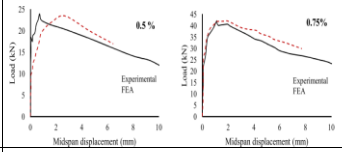
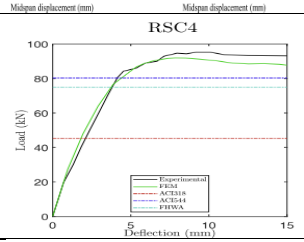
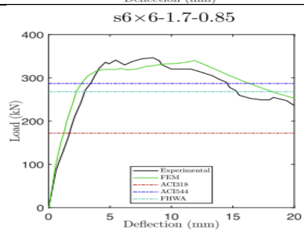
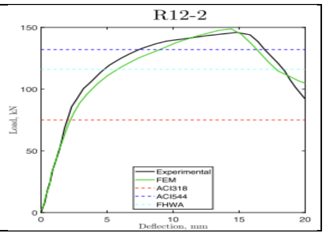
<table>
<tr><td>No.</td><td>investigators</td><td>Type of study</td><td colspan="7">Details of database</td><td colspan="2">Results</td></tr>
<tr><td></td><td></td><td></td><td>Dimensions (mm^3)</td><td>No. of specim-ens</td><td>Type of concrete</td><td>Fiber type</td><td>variable</td><td>L_f (mm)</td><td>V_f (%) or (kg)</td><td>P_{cr} error [kN]</td><td>Max.load error [kN]</td></tr>
<tr><td rowspan="2">[1]</td><td rowspan="2">Hamoda et al.</td><td rowspan="2">ABAQUS</td><td rowspan="2">200×200 ×200mm^3</td><td>12</td><td>NC</td><td>/</td><td>/</td><td>/</td><td>/</td><td>-1.1%~5.4%</td><td>-8.6%~4.5%</td></tr>
<tr><td>12</td><td>SFRC</td><td>hook-end</td><td>/</td><td>/</td><td>/</td><td>-4.4%~7.1%</td><td>-2.6%~9.3%</td></tr>
<tr><td rowspan="2">[2]</td><td rowspan="2">Murthy et al.</td><td rowspan="2">ABAQUS</td><td rowspan="2">200×100 ×1500mm^3</td><td>1</td><td>NSC</td><td>/</td><td>loading</td><td>/</td><td>/</td><td colspan="2" rowspan="2">

Fig. 15. Predicted load – deflection curve of control RC beam compared with average experimental curve.</td></tr>
<tr><td>3</td><td>UHPFRC</td><td>Brass-coaded</td><td>loading</td><td>13mm</td><td>2%</td></tr>
<tr><td rowspan="2">[3]</td><td rowspan="2">Mohsin</td><td rowspan="2">ABAQUS</td><td>150×150 ×550mm^3 (monotonic loading)</td><td>2</td><td>SFRC</td><td>hook-end</td><td>Vf (%)</td><td>30mm</td><td>0%,1%</td><td colspan="2"><table>
<tr><td>Beam</td><td>P_y (kN)</td><td>δ_y (mm)</td><td>P_u (kN)</td><td>δ_u (mm)</td><td>P_{max} (kN)</td><td>δ_{Pmax} (mm)</td><td>$\mu = \frac{\delta_u}{\delta_y}$</td></tr>
<tr><td>Experimental (Vf=0%)</td><td>100.0</td><td>2.15</td><td>127.4</td><td>13.74</td><td>127.4</td><td>13.74</td><td>6.39</td></tr>
<tr><td>FE analysis (Vf=0%)</td><td>95.41</td><td>1.6</td><td>115.0</td><td>11.77</td><td>120.48</td><td>5.80</td><td>7.37</td></tr>
<tr><td>Experimental (Vf=1%)</td><td>122.5</td><td>2.15</td><td>125.1</td><td>19.3</td><td>138.8</td><td>10</td><td>8.98</td></tr>
<tr><td>FE analysis (Vf=1%)</td><td>117.95</td><td>1.6</td><td>132.2</td><td>18.1</td><td>136.36</td><td>5.81</td><td>11.31</td></tr>
</table></td></tr>
<tr><td>150×150 ×550mm^3 (reversed cyclic loading)</td><td>2</td><td>SFRC</td><td>hook-end</td><td>V_f (%)</td><td>30mm</td><td>0%,1%</td><td colspan="2"><table>
<tr><td>Column</td><td>N_c (kN)</td><td>P_y (kN)</td><td>δ_y (mm)</td><td>P_u (kN)</td><td>δ_u (mm)</td><td>P_{max} (kN)</td><td>δ_{Pmax} (mm)</td><td>$\mu = \frac{\delta_u}{\delta_y}$</td></tr>
<tr><td>Experimental</td><td>2.0</td><td>101.8</td><td>2.7</td><td>120.0</td><td>9.41</td><td>120.0</td><td>9.41</td><td>3.49</td></tr>
<tr><td>FE analysis</td><td>1.67</td><td>114.6</td><td>2.7</td><td>126.71</td><td>9.17</td><td>126.71</td><td>9.17</td><td>3.40</td></tr>
</table></td></tr>
</table>

Table 1: Summary of previous experiments on SFRC beams with FEA and Formula

Table 1 (continued-1)

No.	Investigators	Type of study	Details of database: Dimensions (mm^3)	No. of specimen	Type of concrete	Fiber type	Variable	L_f (mm)	V_f(%) or (kg)	Results: Average P_u [kN]	Max.load error [kN]
[4]	Mahmud et al.	ABAQUS	30×150 ×500mm^3	3	UHPFRC	straight		13	2.0%	3.17	11.8%
			60×150 ×500mm^3	3	UHPFRC	straight	d	13	2.0%	10.13	3.6%
			90×150 ×500mm^3	3	UHPFRC	straight		13	2.0%	22.03	4.7%
			120×150 ×500mm^3	3	UHPFRC	straight		13	2.0%	40.73	1.2%
			150×150 ×500mm^3	3	UHPFRC	straight		13	2.0%	61.75	2.7%
[5]	Jin et al.	ABAQUS	200×400 ×2800mm^3	1	SFRC	hook-end	Impact loading and temperatures	30	2%		
[6]	Gao et al.	ABAQUS	120×200 ×1200mm^3	8	SFRC	hook-end	V_f, loading	35	0% to 2%	agree well with the experimental results in terms of the crack patterns, crack numbers, and average crack spacing	
[7]	Shewalul et al.	ABAQUS	150×250 ×3100mm^3	1	SFRC	/	V_f	/	0% to 1.5%		
[8]	Simwanda et al.	ABAQUS	Database from Kahanji	3	UHPFRC	Various shape	V_f	l_f/d_f =6.5	1%,2%, 4%		
			Database from Shafieifar et al.	3			h		2%		

			Database from Yang et al.	3			A_s		2%	

There are some important aspects which should influence the simulation accuracy in FEA with ABAQUS software, such as constitutive relationship to SFRC, selecting of mesh elements to material and other concerning parameters. According to works aforementioned, the concrete plasticity damage model was employed mostly in recent years and which could make FEA results with ABAQUS more accurate than other models (Jin et al., 2018 & Gao et al.,2021). For better results that are as close as possible to practical experimental results, the data from practical experiments which are carried out with beams needed to predict is better than that from other experiments. The average discrepancy could be less than 5% with data belongs to same experiment while the number maybe up to 10% with other data (Mahmud et al., 2013, Murthy et al., 2018 and Jin et al.,2018). Furthermore, more accurate simulation for SFRC beams, in general SFRC are modelled with the brick element C3D8R and steel rebars are model with three dimensional truss elements (T3D2). And the conventional steel reinforcement was embedded in the matrix using the embedded interaction property (Simwanda et al., 2021, Ahmed et al., 2019 and Gao et al., 2021). About the boundary conditions, they have montioned that rigid steel plates were tied on the support and loading points (Gao et al., 2021). The side supports and mid span support were restricted in some direction based on structural behaviors (Yohannes et al., 2021). As for Young's modulus and the Poisson's ratio were usually defined as 45 GPa and 0.22, which were based on current experiment or the other as it is insignificantly for simulation accuracy based on the literature mentioned above (Mahmud et al. ,2013).

Calculation efficiency

Calculation efficiency is concerned by many researchers since it decides how long will the simulation process last. For an identical model, standard of calculation efficiency mainly depends on the number of elements model meshed. It is obviously that the more elements the more accuracy, however, the less efficiency or the longer time the simulation process needs for an identical scale model simulation. Therefore, it should be noted that this question about the number of elements is a balance issue in essence. In general, the parts of model on which stress varies significantly were meshed smaller elements while other parts no need to mesh so small for better calculation efficiency. For investigation of Jin et al. (2018), a mesh size of 10mm was adopted for SFRC and steel rebars while 30mm for other parts. Their choice of element number made the calculation process last 12h which was an acceptable time of simulation while their numerical simulation was enough effective. Mahmud et al. (2013) meshed their beam with d=60mm model, and 5mm, 2.5mm and 1.25mm mesh sizes were selected near the beam center. It was found that the latter two sizes led to virtually identical results. Therefore, their other models were all based on the mesh size 2.5mm and successfully predicted the mechanical performances of SFRC beams with enough accuracy. Moreover, the decision of the steps for loading is as important as the number of elements for the purpose of calculation efficiency and the rules of two aspects are the same basically. This article will not discuss more about it.

CONCLUSIONS

Recent development in FEA simulation of SFRC beams with commercial software ABAQUS have been reviewed. The following concluding remarks are drawn.

1. FEA simulation of SFRC beams with ABAQUS could be accomplished in good agreement between the experimental and numerical predict results.
2. In FEA simulation of SFRC beams with ABAQUS, the concrete plasticity damage model was adopted mostly which was proved to be more accuracy than other concrete models in recent investigations.
3. For identical FEA simulation model with ABAQUS, there is a suitable size of element on parts of model considering calculation efficiency and too small elements are not meaningful to valuable simulation accuracy.
4. Simulation results of SFRC beams with ABAQUS, the discrepancies in general could be less than 10% with suitable data of other researcher's experiment while the number will be limited to 5% with exact average data of experimental results based on the same group of material.

AUTHOR BIOGRAPHY

Hou Zhicheng, a Postgraduate student in Doctor of Philosophy in Civil Engineering (by Research) in Faculty of Engineering and Technology Infrastructure, Infrastructure University Kuala Lumpur. He comes from China, was supervised by Norhaiza Nordin. *Email: 836715392@qq.com*

Norhaiza Nordin, **PhD.** Head of Post Graduate Program (HOPP), Senior Lecturer, Civil Engineering & Construction Division, Faculty of Engineering, Science and Technology, Infrastructure University Kuala Lumpur. *Email: norhaiza@iukl.edu.my*

ACKNOWLEDGEMENT

The authors would like to acknowledge all researchers included in this review paper whose previous researches contributed immensely to the accomplishment of the present research. This research was supported by Infrastructure University Kuala Lumpur and Faculty of Engineering, Science and Technology.

REFERENCES

Ahmed Hamoda, Mohamed Emara, Walid Mansour (2019). Behavior of steel I-beam embedded in normal and steel fiber reinforced concrete incorporating demountable bolted concectors. Composites: Part B, 174 (2019),106996.

A. Ramachandra Murthy, B.L. Karihaloob, P. Vindhya Ranic, D. Shanmuga Priya (2018). Fatigue behaviour of damaged RC beams strengthened with ultrahigh performance fibre reinforced concrete. *International Journal of Fatigue*, 116 (2018), 659-668.

Danying Gao, Zhiqiang Gu, Congjie Wei, Chenglin Wu, Yuyang Pang (2021). Effects of fiber clustering on fatigue behavior of steel fiber reinforced concrete beams. *Construction and Building Materials*, 301 (2021), 124074.

Goran H. Mahmud, Zhenjun Yang, Aram M.T. Hassan (2013). Experimental and numerical studies of size effects of Ultra High Performance Steel Fiber Reinforced Concrete (UHPFRC) beams. *Construction and Building Materials*, 48 (2013),1027–1034.

Liu Jin, Renbo Zhang, Guoqin Dou, Xiuli Du (2018). Fire resistance of steel fiber reinforced concrete beams after low-velocity impact loading. *Fire Safety Journal*, 98 (2018): 24–37.

L. Simwanda , N. De Koker, C. Viljoen (2018). Structural reliability of ultra high-performance fibre reinforced concrete beams in flexure. *Engineering Structures*, 244 (2021),112767.

Sharifah Maszura Binti Syed Mohsin (2012). *Behaviour of fibre-reinforced concrete structures under seismic loading*. London: Imperial Colleage, London.

Yohannes Werkina Shewalul (2021). Numerical and FEA investigation of sectional capacity and moment redistribution behavior of steel fiber reinforced concrete (SFRC) beam. *Heliyon, 7 (2021), e07354.*