STUDY OF SUITABLE SELECTION OF ELEMENT TYPE AND MESH DISCRETISATION FOR FINITE ELEMENT CALCULATION OF STEEL FIBRE REINFORCED CONCRETE ELAVATED SLABS

Nakov Darko [1]

Abstract:

To be able to estimate and optimize capability, reliability and economy of steel fibre reinforced concrete (SFRC) elevated slabs, extensive experimental, numerical and analytical examinations need to be done. In this context it is desirable to be able to evaluate the load bearing behaviour with finite element simulations. On the one hand, such a simulation allows selecting promising systems for experimental examinations and on the other hand it is possible to recalculate experimental examinations for a more specific analysis of the load bearing behaviour. The suitable choice of an element type and a mesh discretisation is a fundamental precondition for an accurate calculation. Hence study needed to be carried out to determine such suitable element types and mesh discretisations for the calculation of SFRC elevated slabs.

Keywords: Steel fibre reinforced concrete elevated slabs; finite element type; mesh discretisation.

1 Introduction

Since the early 60's and 70's the mechanical properties of the steel fibre reinforced concrete were extensively studied. It was proven that it increases mostly the toughness, but also ductility, tensile strength and durability of the structures and is more economical than the conventional concrete. In the early 90's the first full scale experiments were made on elevated slabs. It was proven that they can be a good replacement for the conventional reinforced concrete slabs. Since the material is markedly non-linear after cracking of the concrete matrix, computer performed finite element analysis would be the best choice for determination of the load bearing capacity of steel fibre reinforced concrete elevated slabs. The powerful finite element software package ABAQUS was studied and used in order to find the most suitable element type and mesh discretisation that could be applied for accurate and fast finite element analysis and evaluation of the ultimate load at which a steel fibre reinforced concrete slab would collapse.

[1] Nakov Darko, University "Ss. Cyril and Methodius", Faculty of Civil Engineering, Skopje, Macedonia, nakov@gf.ukim.edu.mk

2 Evaluation of finite element types provided by the software package ABAQUS, suitable for application in slab models mainly subjected to bending

ABAQUS is a powerful engineering simulation program, based on the finite element method, which can solve problems ranging from relatively simple linear analyses to the most challenging nonlinear simulations. It has an extensive finite element library to provide a various set of tools for solving many different problems. Each element in ABAQUS has a unique name, such as S4R, C3D8I or C3D8R. The element name identifies each of the five aspects of an element that characterize its behaviour. The first letter or letters of an element's name indicate to which family the element belongs. For example, S4R is a shell element with four nodes, reduced integrated, according to [3] and [5]. Among all the available families of elements in ABAQUS, the continuum (solid) and shell elements have been used.

Choosing the right continuum element for a specific analysis type can be of crucial importance for the accuracy of the solution obtained. The following continuum element types in regular meshes have been used: C3D8R (eight-node, first order, reduced integrated element), C3D8 (eight-node, first order, fully integrated element), C3D8I (eight-node, first order, fully integrated element enhanced by incompatible modes to improve the bending behavior), C3D20R (twenty-node, second order, reduced integrated element), and C3D20 (twenty-node, second order, fully integrated element).

From the shell element types in Abaqus/Standard only the general-purpose, conventional shell elements have been used. The following shell element types in regular meshes have been used: S4R (four-node, first order, reduced integrated element) and S4 (four-node, first order, fully integrated element).

3 Steel Fibre Reinforced Concrete slender assembly mainly subjected to bending

In order to test and evaluate the chosen finite element types from the previous chapter in bending faster, a simply supported slender beam with span to depth ratio *L/h=25* was chosen. The beam has cross section dimensions *b/h=20/20 cm* and the distance between the supports is *L=500 cm*. The loading scheme is with two forces acting on distance *170 cm* from the supports.

3.1 Analytical solution of the Steel Fibre Reinforced Concrete slender beam

In order to compare the results from the numerical solution of the program system ABAQUS with the analytical solution, the moment-curvature, load-deflection and the load-crack opening displacement curve relations of the SFRC slender beam cross section have been calculated.

For calculation of the point where the first crack occurs, it is assumed that the stress at the bottom of the slender beam reaches the steel fibre reinforced concrete tensile stress f_{ctm}^{f}. For the calculation of the rest of the curve, the used stress and strain distribution, according to [6], is presented in Figure 1. In compression parabola-rectangle stress distribution is assumed, where parabolic stress distribution is up to ε_c = -2 ‰, and up to ε_c =-3.5‰ is rectangle distribution. In tension four-linear stress distribution is assumed as presented in Figure 1. First there is a very sharp peak represented by the value of $f_{ctm}^{f} = 2.56$ N/mm², then there is a constant part from 0.7 N/mm² and at the end slow decrease to 0.55 N/mm².

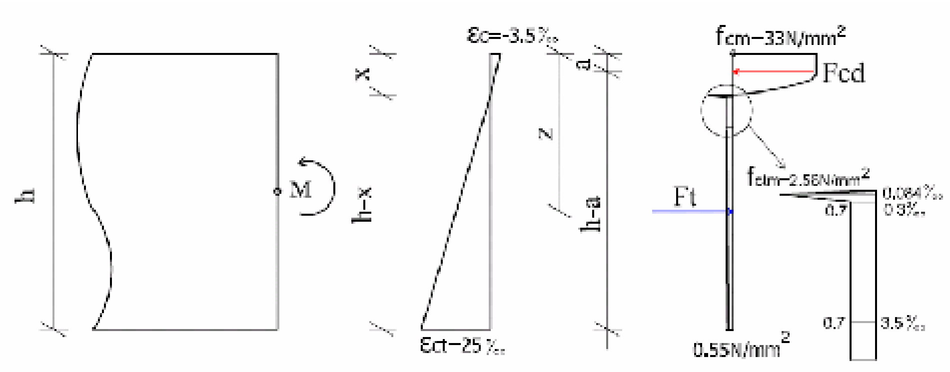

Fig. 1 Stress and strain distribution for SFRC cross section

At the following figure, the load-deflection relation of the SFRC slender beam is presented.

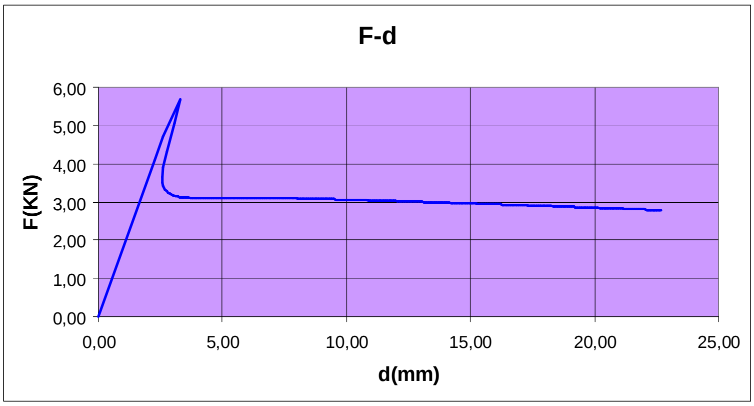

Fig. 2 Relation load-deflection, F-d

3.2 Numerical solution of the Steel Fibre Reinforced Concrete slender beam

The Concrete Damaged Plasticity Material Model from ABAQUS was used, in order to perform the numerical analysis, according to [3] and [5]. As numerical solution technique, full Newton method was used for solving the complex nonlinear equilibrium equations. Loads are applied as a function of time in three steps with different starting time increments, which depends on the need of the nonlinear path to be followed exactly. Because static problems can become unstable because of the nonlinearity, adaptive automatic stabilization was used. This stabilization form is most effective, since the viscous forces with varying damping factor that are added internally to the global equilibrium equations are automatically controlled.

The mesh discretisation with continuum elements of the assembly was made using three different meshes, course mesh M1, medium mesh M2 and fine mesh M3, as presented in Table 1. The three meshes are chosen regarding the number of elements through the thickness of the beam.

Tab. 1 Three different mesh types for continuum element types

Mesh type	Element length (mm)	Elements through thickness	Total number of elements
M1	100	2	216
M2	50	4	1728
M3	33.33	6	5832

The mesh discretisation with shell elements of the assembly was made using the medium mesh M2 for X-Y direction and using two different integration rules, Gauss and Simpson, through the thickness of the shell (Z direction), each with three different number of integration points, as presented in Table 2. In total we have six different mesh types which represent the density of the integration points through the thickness of the shell.

Tab. 2 Six different mesh types for shell element types

Mesh type	Element length (mm)	Type of integration through thickness	Number of integration points through thickness
G3_M2	50	Gauss integration	3
G5_M2	50	Gauss integration	5
G7_M2	50	Gauss integration	7
S5_M2	50	Simpson integration	5
S7_M2	50	Simpson integration	7
S9_M2	50	Simpson integration	9

3.3 Interpretation and discussion of the results from the numerical solution and comparison with the analytical solution

The previously mentioned meshes in Table 1 and Table 2 have been evaluated with all of the chosen continuum or shell finite element types. Here the results only for the continuum element C3D8 and the shell element S4 are presented.

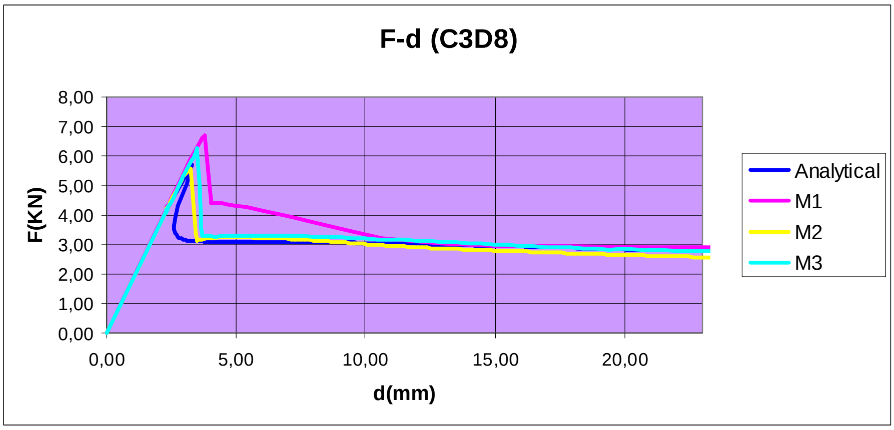

Fig. 3 Comparison between analytical and numerical solution (Mesh M1, M2 and M3) of the load-deflection relation for the solid element C3D8

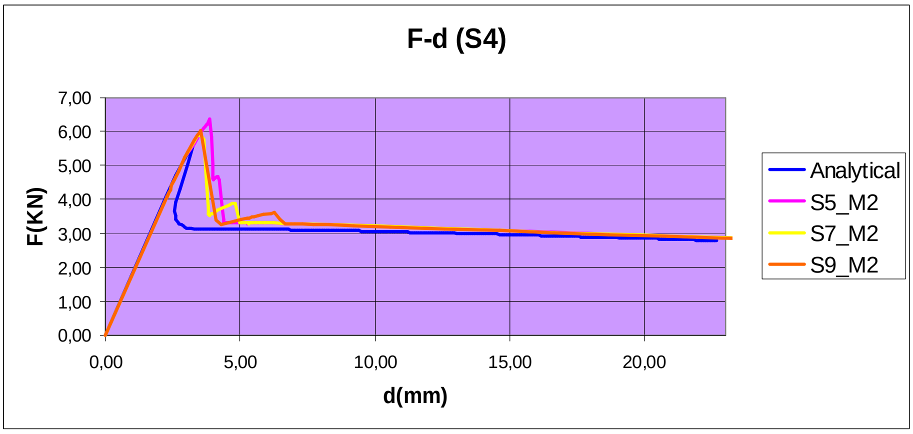

Fig. 4 Comparison between analytical and numerical solution (Mesh S5_M2, S7_M2 and S9_M2) of the load-deflection relation for the shell element S4

For the chosen and examined solid element types the following was concluded:

- If we want to model the localized plastic strain region in a steel fibre reinforced concrete slender beam with solid element types, the course mesh M1 gave poor results and should not be used, and the fine mesh M3 increase the time and memory and should be used only when there is no other opportunity;
- The medium mesh M2 was chosen as the most realistic one, in view of accuracy and calculation time;
- If the linear reduced integration element C3D8R is chosen to be used, then the fine mesh M3 is the only opportunity, or even finer meshes may be needed to avoid the hourglassing effect;
- Although the incompatible mode element C3D8I used in linear elastic analysis has shown the best efficiency, in our specific non-linear material model has shown a big sensitivity;

- The second order fully integrated element C3D20 is not recommended due to the fact that gave less accurate results than the corresponding reduced integration element C3D20R because of the stiffer behaviour during the calculation procedure.
- The only element types that provided good results with the medium mesh M2 are the fully integrated linear solid element C3D8 and the reduced integrated quadratic element C3D20R;
- Both element types, C3D8 and C3D20R, have the same number of integration points (8), but C3D20R has more nodes, which means that the calculations take more time and computer memory;
- Finally, element C3D8 was chosen like the most suitable one from the solid element types, but at least four elements through the thickness should be used (mesh M2), to avoid the effect of shear locking.

From the chosen and examined shell element types the following was concluded:

- From the load-deflection relations it was concluded that only the mesh type that use Gauss integration with three integration points through the thickness of the shell (G3_M2), has shown bad results;
- If Gauss integration is to be used, at least five integration points through the thickness of the shell are required to provide satisfactory results.
- Due to the fact that using Simpson integration rule we can get the results at the top and bottom surface of the shell, and the fact that the number of integration points in ABAQUS is not limited, Simpson integration was chosen like more suitable one;
- Because we have to model very complex material behaviour, more integration points through the thickness are required to represent the material correctly;
- Therefore, the Simpson integration rule with nine integration points (S9_M2) was chosen like the best mesh discretisation for the shell element types;
- There is no great difference between the linear fully and reduced integration elements, S4 and S4R, in the accuracy of the results, but also, in view of time needed for the calculations. Therefore the more confident fully integrated element S4 was chosen like the most suitable one from the shell element types.

4 Steel Fibre Reinforced Concrete elevated slab

The analysed slab is square simply supported by four columns with similar span to depth ratio as the previous analysed beam, L/h=26. The thickness of the slab is 20 cm and the dimensions are Lx/Ly=520 cm. The loading scheme is a concentrated force F in the middle of the slab.

4.1 Analytical solution of the Steel Fibre Reinforced Concrete elevated slab

In order to find out the ultimate moment which we need for later determination of the load bearing capacity, the moment-curvature relation for the SFRC elevated slab cross section was calculated. For calculation of the point where the first crack occurs, it is assumed that the stress at the bottom of the elevated slab reaches the steel fibre reinforced concrete tensile stress f^{f}_{ctm}. For the calculation of the rest of the curve, the same stress and strain distribution as for the slender beam, according to [6], is used and is presented in Figure 1. After the first crack happen, at value of the moment of 17.1 kNm/m', the moment still increases up to a value of 24.19 kNm/m' at ε^{f}_{ct} = 0.2 ‰. After that it suddenly decreases and after $\varepsilon^{f}_{ct} \cong$ 1.3 ‰ it decreases very slowly from 13.52 kNm/m' up to 11.83 kNm/m' at ε^{f}_{ct} = 25 ‰. In this range should be the ultimate load at which the slab will collapse. Therefore an average of 12.68 kNm/m' was used like an ultimate moment for calculation of the ultimate load. The analytical calculation of the load bearing capacity of the steel fibre reinforced concrete elevated slab was done using the Yield Line Theory and a value of the ultimate load of 50.72 kN have been obtained.

4.2 Numerical solution of the Steel Fibre Reinforced Concrete elevated slab

For implementation of the steel fibre reinforced concrete material the Concrete Damaged Plasticity Material Model from ABAQUS was used. In order to perform displacement controlled full Newton iteration, 30 mm displacement was prescribed in the centre of the slab. The load is applied as a function of time in two steps with different starting time increments, which depends on the need of the nonlinear path to be followed exactly.

Tab. 3 Three different mesh types for shell element type S4

Mesh type	Element length (mm)	Type of integration through thickness	Number of integration points through thickness
S9_M1	400	Simpson integration	9
S9_M2	200	Simpson integration	9
S9_M3	100	Simpson integration	9

Each of the three different mesh types for the shell element type S4 are made with the chosen Simpson integration rule from the previous chapter with nine integration points through the thickness of the shell. The meshes are presented in Table 3.

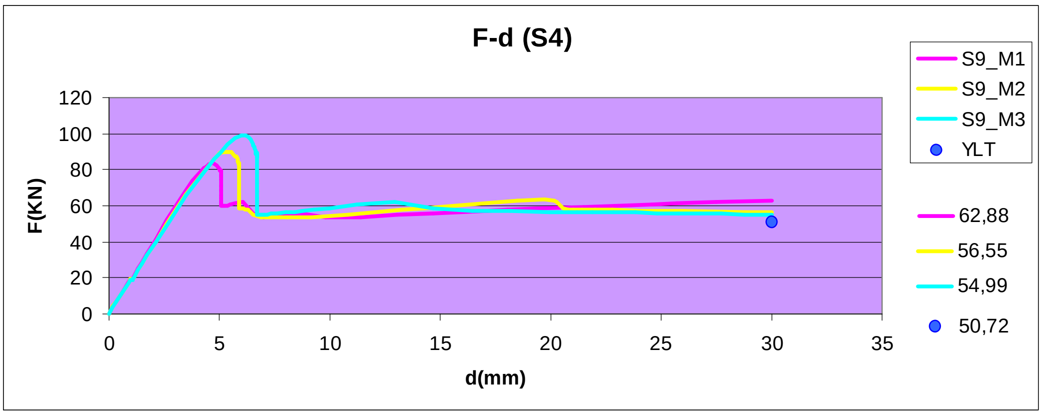

Fig. 5 Comparison between the numerical solutions of the load-deflection relation for the shell element S4 and the ultimate load by the Yield Line Theory (YLT)

In the previous Figure 5, the comparison of the load-deflection relation between the numerical solutions and the analytically calculated ultimate load by the Yield Line Theory is presented.

5 Conclusions

From the presented load deflection relations in Figure 5 it can be noticed that only mesh type S9_M1 lead to big overestimation of the ultimate load. Therefore the conclusion is that element length of 400mm is too big to represent the formation of the yield lines. Already with mesh type S9_M2 the difference with respect to the Yield Line theory is 11.5% and with mesh S9_M3 is only 8.42%. If we take into account that the ultimate moment was taken as an average value, which means that few percents mistake is possible in the Yield Line Theory, than we can come even close to the solution.

A conclusion can be made that for this static system, between the numerical calculations and the Yield Line Theory, a close agreement was found. Recommended span to depth ratios are around 25.

We can conclude that structures with possibility of moment redistribution are favorable for steel fibre reinforced concrete material. Since the steel fibre reinforced concrete provides post-cracking load bearing capacity, with design at the ultimate limit state the material properties can be taken into account.

For numerical finite element calculation, linear fully integrated shell element S4 with element length of 100 mm provides solution for the ultimate load that is in close agreement with the Yield Line Theory.

Aknowledgements

Since this paper is a part of my Master thesis, I would like to thank my mentor Dipl.-Ing. Lars Goedde for the constant support and mentoring during the making of the thesis. Without his help the making of the thesis would not be possible. The support of Univ.-Prof. Dr.-Ing. Friedhelm Stangenberg and the advices of Dr.-Ing. Mathias Strack are gratefully acknowledged.

References

J.Edgington : Steel fibre reinforced concrete

Test and design methods for steel fibre reinforced concrete - Brite Euram BRPR-CT98-0813-Project nr: BE 97-4163

Abaqus Theory Manual version 5.6 - Hibbitt, Karlsson & Sorensen, Inc.,1996

Abaqus/Standard User's Manual Volume I, II & III version 5.6 - Hibbitt, Karlsson & Sorensen, Inc., 1996

Abaqus/Standard version 6.7 online documentation

DAfStb-Richtlinie: Stahlfaserbeton (23.Entwurf) - Ergaenzung zu DIN 1045, Teile 1 bis 4, Beuth Verlag, Berlin, 2005