Malaysian Journal of Civil Engineering 19(2) :91-103 (2007)

QUASI-STATIC ANALYSIS OF COMPOSITE SLAB

Redzuan Abdullah[1,*], Vidal P. Paton-Cole[2], W. Samuel Easterling, F.ASCE[3]

[1]*Faculty of Civil Engineering, Universiti Teknologi Malaysia*
[2]*Graduate Student, Faculty of Civil Engineering, Universiti Teknologi Malaysia*
[3]*Professor and Assistant Dept. Head, The Via Dept. of Civil and Environmental Engineering, Virginia Tech, Blacksburg, VA 24061, USA*

[*] Corresponding Author*: Redzuan@utm.my*

ABSTRACT: This paper presents a method for modeling and analyzing steel deck-concrete composite slabs to obtain quasi-static response using non-linear dynamic finite element analysis. A three-dimensional finite element model of steel-concrete composite slab was developed and analyzed using ABAQUS/Explicit module software. The main considerations included in the model were tensile brittle cracking of concrete, horizontal shear bond behaviour between the concrete and the steel deck and the analysis control. The horizontal shear bond characteristic of the slab model was deduced from bending test data. The finite element modelling and analysis method presented here for composite slabs which failed by a combination of concrete tensile brittle cracking and horizontal shear slip described here was found to be accurate compared to bending test data.

Key words: *Composite Slabs; Quasi-Static Analysis; Non-linear Finite Element Analysis*

ABSTRAK: Kertas kerja ini membentangkan satu kaedah bagi memodel dan menganalisis papak rencam dek keluli-konkrit bagi mendapatkan gerak balas kuasi-statik dengan menggunakan analisis unsur terhingga dinamik tak lelurus. Sebuah model unsur terhingga tiga-dimensi bagi papak rencam dek keluli-konkrit telah dibangunkan dan dianalisis dengan menggunakan modul perisian ABAQUS/Explicit. Pertimbangan utama yang telah diambil kira dalam model berkenaan adalah retakan rapuh tegangan pada konkrit, kelakunan ikatan ricih ufuk antara konkrit dengan dek keluli, dan kawalan analisis. Ciri ikatan ricih bagi model papak telah diterbitkan daripada data ujian lenturan. Kaedah pemodelan dan analisis unsur terhingga yang diterangkan di sini untuk papak rencam dek keluli-konkrit yang gagal secara gabungan antara retakan rapuh tegangan konkrit dengan gelincirin ricih ufuk telah berjaya menghasilkan gerak balas papak rencam yang tepat berbanding dengan data ujikaji lenturan.

Kata kunci: *Papak Rencam; Analisis Kuasi-Statik; Analisis Unsur Terhingga Tak Lelurus*

1.0 Introduction

Present design codes for steel-concrete composite slabs require that the design parameters must be obtained from full scale bending tests (ASCE, 1992; CSSBI, 1996; Eurocode 4, 2001). Design methods based on the test data are expensive and time consuming. Because of this reason, modeling and analysis of composite slabs using finite element (FE) method has been the subject of much research. This has been driven by the significance of providing a cheap and accurate tool to predict the behavior and performance of composite slabs.

Several FE models have been proposed by past researchers to account for the complex morphology of the steel deck-concrete interaction in composite slabs. Daniels and Crisinel (1993) developed a special purpose FE procedure using plane beam elements for analyzing single and continuous span composite slabs. The procedure incorporated nonlinear behavior of material properties, additional positive moment reinforcements, a load-slip property for shear studs and a shear interaction property between the concrete and the steel deck. The shear interaction property was obtained from a pull out test. An (1993) studied the behavior of composite slabs with 2D non-linear FE model using ABAQUS/Standard module software. The steel sheeting and the concrete were modeled as 2-node Timoshenko beam element. The interaction between the steel sheeting and the concrete slab was modeled with spring elements plus a set of imposed equations between degrees of freedom of the beams and spring elements. The spring property was obtained from a block bending test. Veljkovic (1996) performed 3D FE analysis using DIANA to study the mechanics of force transfer between the steel sheeting to the concrete in composite slabs. Based on tensile tests of embossed and flat parts of the sheeting, different uniaxial stress-strain relationships were used for the web and the flange. The shear interaction between the sheeting and the concrete was modeled using a nodal interface element and its property was obtained from a push test. Widjaja (1997) performed FE analysis of composite slabs based on the model similar to that used by An (1993). Slabs with shear stud and weld anchorage at the supports were analyzed. The force-slip relationship for the interface element was obtained from elemental tests similar to the pull out test used by Daniels (1988) and the end anchorage was obtained from a modified composite beam push off test. Abaqus (2002) outlined a quasi-static procedure for analysis of a simply supported square reinforced concrete slab under a point load.

This paper proceeds with providing an accurate modeling and analysis method for obtaining load-deflection behavior of steel deck-concrete composite slab under bending. Quasi-static analysis using ABAQUS/Explicit Dynamics 6.3 was implemented. The explicit dynamic solution procedure was chosen because it is most accurate in applications where brittle behaviour dominates (Abaqus, 2002). The brittle cracking behaviour is demonstrated by concrete tensile cracking in composite slabs. A three-dimensional non-linear finite element model of steel deck-concrete composite slab was developed and analyzed. The main considerations included in the model were tensile brittle cracking of the concrete, horizontal shear bond interaction between the concrete and the steel deck and the analysis control. The shear bond characteristic of the slab was deduced from a bending test.

2.0 Modelling Method

2.1 Structural model

A strip of 305 mm wide slab that was considered in this study is shown in Figure 1(a) and the cross section dimensions are in Figure 1(b). Because of the symmetric condition, only one-quarter of the strip as shown in Figure 1(c) was considered in the FE model. The model was similar to a two point load bending test specimen as shown in Figure 2. The detail descriptions and validation of the tests are available in Abdullah (2004). The steel deck details are shown in Figure 3.

The concrete slab was modeled with 8-node linear brick, reduced integration elements (C3D8R) while the steel deck was modeled with 4-node doubly curved general purpose shell, elements (S4R). The steel deck elements located underneath the concrete slab elements were offset from the bottom of concrete elements at a distance half of the steel sheeting thickness. Figure 4 depicts the slab model with the steel sheeting offset away from the concrete element for clarity.

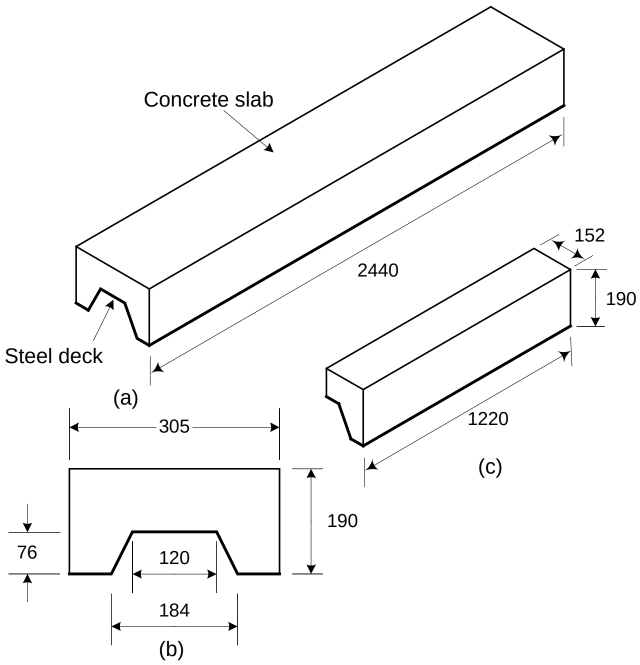

Figure 1: Composite slab dimensions (mm). (a) One deck corrugation, (b) Cross section, (c) One-quarter model

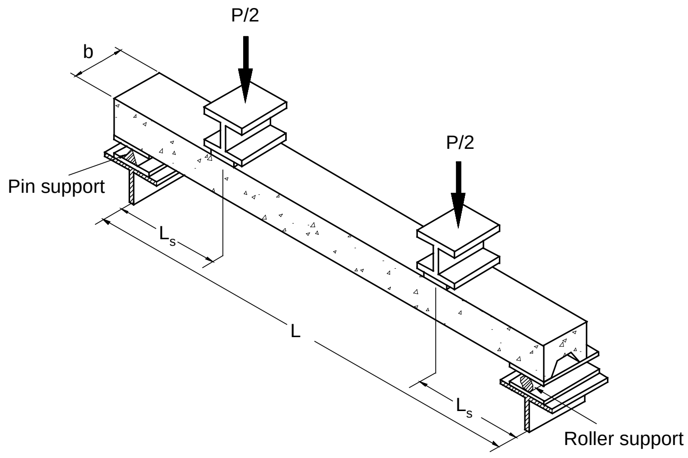

Figure 2: Isometric view of small scale (narrow) test setup (Abdullah, 2004)

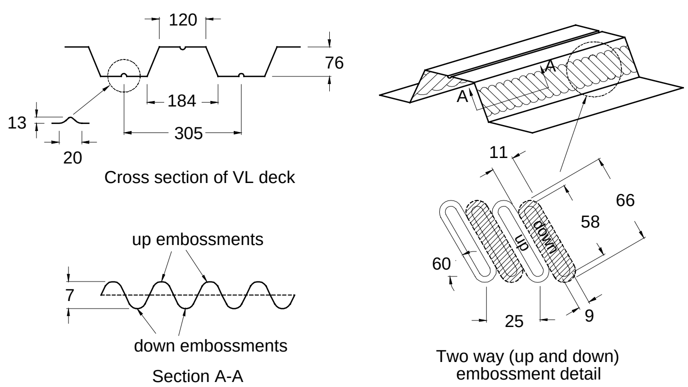

Figure 3: Deck geometries, in mm

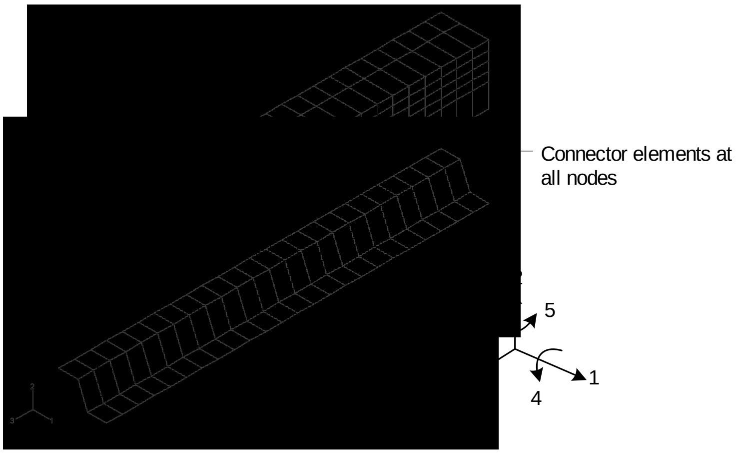

Figure 4: Finite Element model

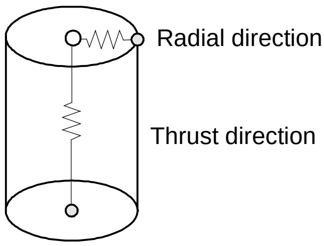

Figure 5: Radial-thrust connector element

The interaction between the concrete and steel deck was modelled with radial-thrust type connector element (CONN3D2) as shown in Figure 5. The connector elements were used to connect the concrete nodes to steel sheeting nodes that were closest to each other. The radial component was orientated in the longitudinal direction of the slab to represent the horizontal shear bond while the thrust component was for the vertical interaction. The use of radial-thrust element enabled the behavior for the horizontal and vertical displacements of the slab to be specified separately.

A roller support was provided at 100 mm from the slab end and was applied to the nodes of the bottom flange of the steel sheeting. In order to simulate the continuity of the slab in the transverse direction, nodes at the side faces of the concrete along the longitudinal length were fixed in the 1-direction (refer Figure 4) and nodes of the edges of steel sheeting along the longitudinal length were fixed in the 1-direction, and 5- and 6-rotations. To achieve the half span model, the concrete nodes at the mid span coordinate were fixed in the 3-direction, and the steel sheeting nodes were fixed in the 1-direction, and 4- and 5-rotations.

2.2 Concrete Material Model

All concrete elements except those in the top layer were modeled with BRITTLE CRACKING function. This material model considered the concrete behavior to be predominantly governed by tensile cracking. The model also assumed that the compressive behavior was always linear elastic. This assumption was consistent with the observation of slab behavior during the bending test, where the concrete did not fail by crushing but rather separated by excessive tensile cracking underneath one of the point load due to slip. The same observation was reported by Luttrell (1987). The concrete elements in the top most layer were assigned a linear elastic property where no cracking was allowed. This was done to avoid convergence problems where the model could become numerically unstable before reaching ultimate load if the crack was allowed all the way up to the top most layer.

Table 1. Concrete mechanical and brittle cracking properties used in the FE model.

Concrete Properties	Values
Density	2400 kg/m^3
Elasticity modulus	24.8 GPa
Poisson ratio	0.2
Cracking failure stress	2.07 MPa
Mode I fracture energy	73.56 N/m
Direct cracking failure displacement	1.27x10^{-5} m
2 segments tension stiffening model	Remaining direct stress, Direct cracking displacement 2.07 MPa, 0 mm 0.62 MPa, 0.022 mm 0 MPa, 0.140 mm
Post cracking shear behaviour model	Power law with 2.0 power factor 0.4 % maximum crack opening strain for coarse mesh. The value was adjusted proportionately according to characteristic mesh size for models with different mesh sizes.

The concrete properties and brittle cracking parameters were assumed based on various literatures and the final values were fixed after an admissible result was obtained. The final concrete properties applied in the model are given in Table 1.

2.3 *Steel Material Model*

Veljkovic (1994) reported that the indentations pressed in the web had reduced the effective yield stress and the elastic modulus to 47% of the original values of the flat sheet. Strain gage readings obtained from the bending tests confirmed that at some point during the load application, the bottom flange of the steel deck yielded (Abdullah and Easterling, 2003). In this study, both linear elastic and elastic-perfectly plastic behavior that uses Von Misses yield surface were tried. Based on Veljkovic's finding, several reduced yield stress and modulus of elasticity values for the web elements were tried.

Table 2. Steel properties used in the FE model

Steel properties	Values
Density	7800 kg/m^3
Elastic modulus (flanges)	203.4 GPa
Yield stress (flanges)	345 MPa
Elastic modulus (web)	101.7 GPa
Yield stress (web)	173 MPa

An elastic-perfectly plastic model with web yield stress and modulus of elasticity reduced by 50% of the flange values had produced results that correlated well with bending test data. The final properties of the steel sheeting used in the model are given in Table 2.

2.4 *Horizontal Shear Bond Model*

The horizontal shear interaction between the concrete and the steel deck was modelled by the connector element as discussed in Section 2.1. The interaction behaviour was represented by the horizontal shear stress-end slip relationship as shown by the curves in Figure 6. This behaviour was derived from data obtained from the bending tests (Figure 2) using *force equilibrium method* as detailed out in Abdullah (2004). The values in the form of shear bond forces versus displacements calculated from the curve were assigned to the radial component of the connector elements. The vertical interaction was not considered in the model assuming that its effect was implicitly present in the horizontal shear stress-end slip curve. This assumption was made due to the fact that the horizontal shear stress-end slip curve was derived directly from the bending tests. For the same reason, the frictional resistance at the support was also excluded in the model. A relatively stiff value was assigned to the thrust component of the connector elements so that the concrete and the steel elements would not overlap when the load was applied.

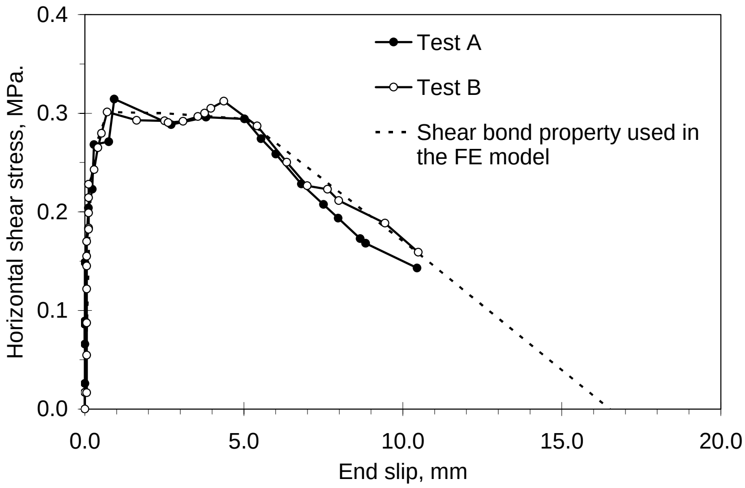

Figure 6: Horizontal shear bond stress versus end slip curve derived from bending tests

3.0 Loading and Analysis Control

Before performing the quasi-static analysis of the slab, an Eigenvalue frequency analysis which is available in ABAQUS/Standard module was conducted to obtain natural frequency and hence natural period of the model. The natural period was used as an indicator for specifying the time step period in the quasi-static analysis.

In performing the quasi-static analysis, the load was specified by a prescribed maximum vertical displacement at concrete top fiber nodes where the actual line load was applied. Because ABAQUS/Explicit is a dynamic analysis program, and in this case a quasi-static solution is desired, the prescribed displacement was increased slowly enough to eliminate any significant inertia effect. The displacement was increased linearly using a smooth amplitude function over a time step period of five to ten times longer than the natural period.

Before any result was accepted, kinetic energy was compared with internal energy of the whole model throughout the analysis period to give a general indication whether the quasi-static solution was obtained. The kinetic energy was chosen as an indicator for the quasi-static response based on the conservation of energy principle. The principle states that the time rate of change of kinetic energy and internal energy for a fixed body of material is equal to the sum of the rate of work done by the external forces. The quasi-static response was ensured by keeping the kinetic energy level due to the movement of the model to below 5% of the internal energy at any instance during the time step period. In this way, the external work done by the load is balanced mostly by the internal energy of the whole structure, as is actually happens in static analysis. Limiting the kinetic energy level was attained by adjusting the combination of time step period and the total displacement at the loading nodes accordingly, and monitoring the plots of the kinetic and internal energy with respect to time step period. Longer loading periods can reduce dynamic effects but may increase analysis time significantly. An optimum analysis period was obtained after several trials.

4.0 Result and Discussion

At the initial development of the model, the analyses were carried out using coarse mesh shown in Figure 4 to determine a good combination of material properties and to obtain an admissible quasi-static solution. For each trial combination of material properties, the analyses were performed by varying the time step period and total vertical deflection at

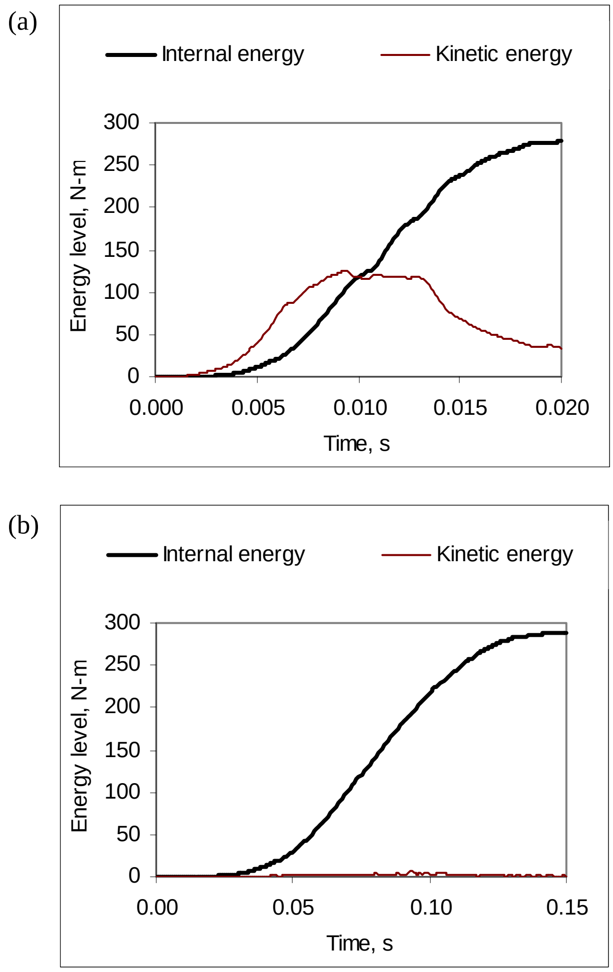

Figure 7: Energy level of the whole model for analysis with step time equal to (a) 1x natural period (b) 8x natural period

loading nodes. For each result, internal and kinetic energies of the whole model were plotted. Figure 7(a) depicts a typical internal and kinetic energy levels during the analysis period for the dynamic response while Figure 7(b) is for the quasi-static response. The corresponding reaction force versus mid-span deflection curves are shown in Figure 8(a) and Figure 8(b) respectively.

.

(a)

(b)

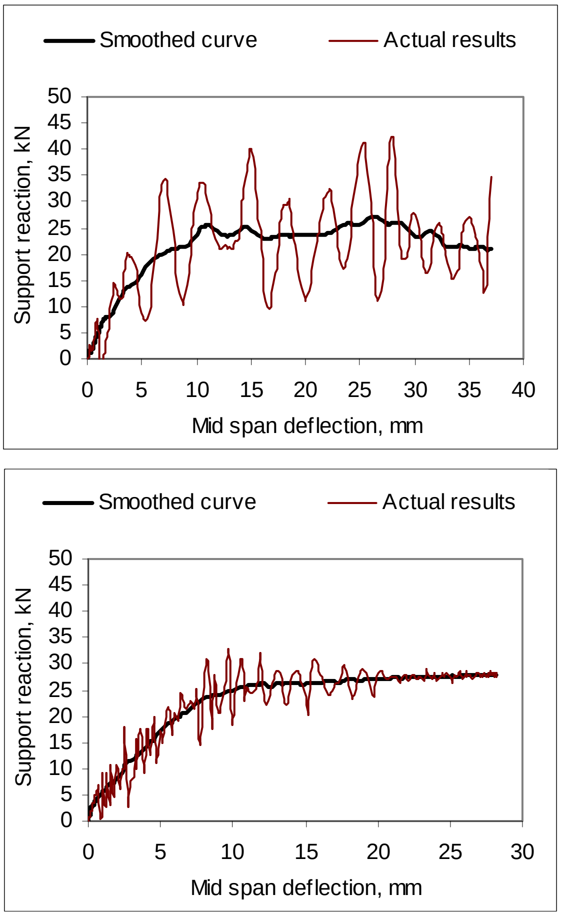

Figure 8: Typical analysis results, (a) Dynamic response, (b) Quasi-static response

As mentioned before, the quasi-static response was assured by keeping the kinetic energy level below 5% of the internal energy level. However, because ABAQUS/Explicit is a dynamic analysis program, oscillating response was inevitable even in the quasi-static solution. The oscillation magnitudes became larger for smaller step time period but reduced as the step time period increased. The quasi-static results were generally obtained when the time step periods were greater by 5 times or more than the slab natural period. The results were then smoothed using the smoothing function available in ABAQUS/CAE module to eliminate the oscillation effect. Figure 8(a) and Figure 8(b) also depict the smoothed curves obtained from the actual dynamic results

The quasi-static analysis procedure was tried further with three element sizes designated as coarse, medium and fine, as shown in Figure 9 using the same material properties listed in Tables 1 and 2 to illustrate mesh sensitivity. The connector elements for the fine mesh were located at the same locations as for the coarse and medium mesh models, that is at 50 mm interval along the longitudinal length. Results of the analyses using three different mesh sizes were plotted together with the test data for comparison. The plot is shown in Figure 10. The graphs show that the models were insensitive to the element size.

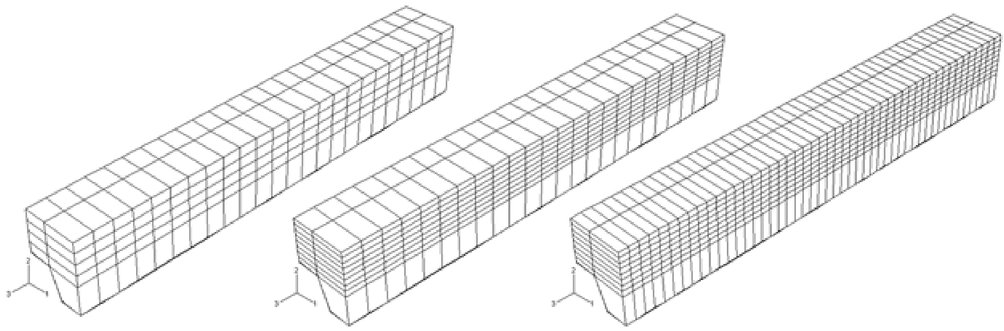

Figure 9: Coarse, medium and fine mesh models

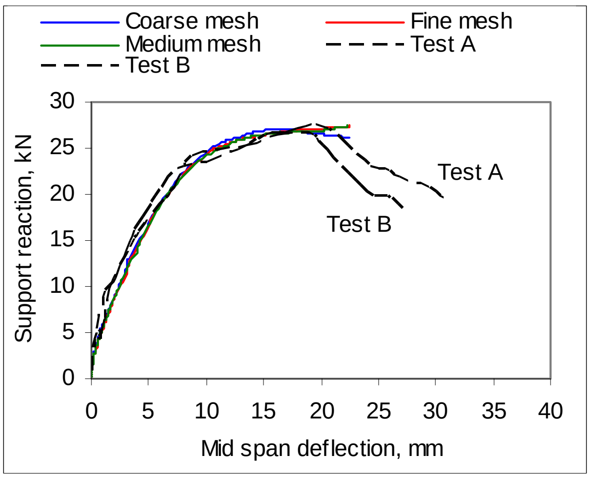

Figure 10: Results of analyses using coarse, medium and fine meshes and comparison with test data

However, as observed during the analysis, the coarser mesh model responded more sensitively to the change of time step period. This was reflected by greater vibration magnitudes for the coarser mesh as compared with the vibration magnitude of the finer mesh model.

5.0 Conclusion

An accurate quasi-static analysis method for modeling and analyzing composite slabs is proposed. ABASQUS/Explicit dynamic package was used in order to take advantage of the concrete brittle cracking model which is only available in this software package. The brittle concrete cracking model is sufficient for modelling concrete tensile cracking that occurs in the composite slabs. Connector elements were used to represent the shear interaction behavior between the concrete and the steel deck. The property of the connector element was derived from bending test and was found to be suitable for use in the finite element model. The quasi-static analysis method presented in this study is capable of predicting the load-deflection behavior and the ultimate load of composite slabs accurately. Thus the method can be used for analyzing composite slabs with brittle concrete cracking. The method also can be used as an alternative to static analysis. The results obtained from the analysis can be used to predict the behaviour of composite slabs without resolving to full scale tests as required by major design specifications.

References

Abaqus Online Documentation, Version 6.3-1 (2002) Hibbit, Karlsson and Sorenson Inc. Pawtucket, R.I.

Abdullah, R. and Easterling, W.S. (2003) *Structural evaluation of new Vulcraft composite deck profile: Phase II*, Report No. CEE/VPI-ST03/01, Virginia Polytechnic Institute and State University, Blacksburg, Virginia.

Abdullah, Redzuan (2004) Experimental Evaluation and Analytical Modeling of Shear Bond in Composite Slab", PhD Dissertation, Virginia Polytechnic Institute and State University, Blacksburg. USA.

An, L. (1993) Load Bearing Capacity and Behaviour of Composite Slabs with Profiled Steel Sheet, Phd Dissertation, Chalmers University of Technology, Sweden.

ASCE. (1992). *Standard for the Structural Design of Composite Slabs. ANSI/AASCE 3-91*, American Society of Civil Engineers, New York.

CSSBI. (1996) *Standard for Composite Steel Deck, CSSBI 12M - 96*, Canadian Sheet Steel Building Institute.

Daniels, B. J., and Crisinel, M. (1993) Composite Slab Behavior and Strength Analysis. Part I: Calculation Procedure. *Journal of Structural Engineering*, 119(1-4), 16-35.

Daniels, B. J. (1988) Shear Bond Pull-out Tests for Cold-Formed-Steel Composite Slabs. ICOM -Construction Metallique, Department de Genie Civil, Ecole Polytechnique Federale de Lausanne.

Eurocode 4. (2001) *Design of Composite Steel and Concrete Structures - Part 1.1: General Rule and Rules for Buildings, EN 1994-1.1:2001, Draft No. 3*, European Committee for Standardization, Brussels.

Luttrell, L. D (1987) Flexural Strength of Composite Slab. *Composite Steel Structures – Advances, Design and Construction,* Narayanan, R., Ed., Elsevier, London, 106-115.

Veljkovic, M. (1994) Interaction between Concrete and Sheeting in Composite Slabs. *S-971 87 LULEA*, Lulea University of Technology, Division of Steel Structures, Lulea, Sweden.

Veljkovic, M. (1996) Behaviour and Resistance of Composite Slabs, Phd Thesis, Lulea University of Technology, Lulea, Sweden.

Widjaja Budi Ryanto (1997) Analysis and Design of Steel Deck-Concrete Composite Slabs, Ph.D. Thesis, Virginia Polytechnic Institute and State University, Blacksburg. USA.