CHAPTER 4

STRENGTH AND STIFFNESS PREDICTIONS OF COMPOSITE SLABS BY FINITE ELEMENT MODEL

4.1. General

Successful use of the finite element method in many studies involving complex structures or interactions among structural members has been one of the motivations for applying the method in this study. To compare with simple mechanical models discussed in the previous chapter, finite element models may offer more accurate analyses because of the ability to model the material and interaction of each part of the system in more detail. Further, the response history of virtually any part of the model can be obtained. In this method, element and material model types play an important role for the entire analysis. Selection of element and material model types for the analysis is based on the structural system and any specific need or emphasis of the study.

In this study, because the main concern is behavior of one-way composite slabs with a large ratio of length to the cross sectional dimensions in a typical width of the slab, then the choice of beam and spring elements for a finite element model is the most effective one. The model is similar to the one proposed by An (1993) with modifications such as the inclusion of end anchorages and a concrete fracture model for concrete in tension. With this concrete fracture model, the mesh sensitivity problem in finite element analysis involving concrete (brittle) material can be removed (Fracture 1992; Karihaloo 1995). Descending curves of end anchorages and shear bond interaction are also included. ABAQUS is used to conduct the analyses.

4.2. Review of Finite Element Method for Composite Slabs

Due to the complex nature of interactions within composite slab systems, finite element modeling has become a powerful tool in predicting the slab strength and stiffness. The power rests in the ability to locally model each different part or interaction of the system and systematically integrate contributions of those parts or interactions to represent the whole system. For composite slabs, various models have been proposed. The selection of model types depends on the physical system of the slabs and specific need of the study.

Daniels et al. (1989, 1990), Ren and Crisinel (1992) and Ren et al. (1993) used plane-beam elements to model one-way composite slabs. Special ten-degree of freedom beam elements that can take into account nonlinear slip behavior between the steel deck and concrete slab was used. For this purpose, a special finite element code was developed.

By using ABAQUS, a commercial general-purpose finite element code, two-node plane Timoshenko beam elements were used by An (1993) for one-way slab systems. Two series of beam elements were generated, one for the concrete slab and the other for the steel deck. Shear bond interaction was modeled by using series of spring elements and additional set of equations to the stiffness equations to prescribe imposed relations among the degree of freedoms of the spring, concrete beam and steel deck beam nodes.

Three dimensional brick elements were used for a two-way composite slab system. Some difficulties concerning numerical convergence was reported in the 3D model (An 1993). The problem was thought to be due to mesh sensitivity in relation to the tension-stiffening model of the concrete material. Because of this problem, the concrete material model was replaced by two different J_2 (metal) plasticity models, each of which is representing the tension and compression parts of the concrete separately.

Other 3D models using brick elements were proposed by Veljkovic (1993, 1994, 1996) and Oloffsson et al. (1994). In their study, DIANA, a general-purpose finite element code was used. It was reported that some trials for concrete tension stiffening functions were needed in some cases before a stable numerical result can be obtained.

4.3. Finite Element Model

4.3.1. Structure Model

A simply supported beam configuration is chosen as a typical model of the system. In the case with continuous deck over an interior support, a rotational spring element is added to the continuous end. The stiffness of this spring represents an elastic rotational stiffness of the adjacent span at the common support. This type of configuration (simply supported beam) is based on observations during experimental tests. Because of the absence of negative reinforcement over interior supports, negative cracks along these supports were developed at a relatively low load level. Therefore, the assumption that the concrete slab is discontinuous over the interior support will not have any significant effect to the analysis.

Two series of Euler-Bernoulli beam elements with 12 in. typical length were generated. One series is for the concrete slab and the other is for the steel deck. Only a single typical longitudinal *slice* of the slab is considered in the model. Vertical nodal displacements of these two series of beam elements are forced to be the same. This is based on previous study (An and Cederwall 1994) which concluded that the effect of vertical separation between the two parts is minor.

End anchorages and shear bond interactions at the interface of the concrete and steel deck are modeled by using horizontal spring elements. In the case of the shear bond interaction, the spring elements are placed along the slab. One end of each spring element is attached to the steel deck beam element and the other to the concrete beam element. Both are at the steel deck centroid elevation. This means that the attachment of the spring elements to the concrete beam element is *not* at the centroid of concrete beam elements. In ABAQUS, this can be assigned by using the EQUATION option in which the magnitude of a certain degree of freedom can be made equal to scalar multiplications of any other degree of freedoms. This compatibility condition is shown schematically in Fig. 4-1 and can be expressed as:

$$y_c = y_d \sin\theta \cong y_d\,\theta \tag{4-1}$$

$$y_s = u_1^d - u_1^c + y_d\,\theta \tag{4-2}$$

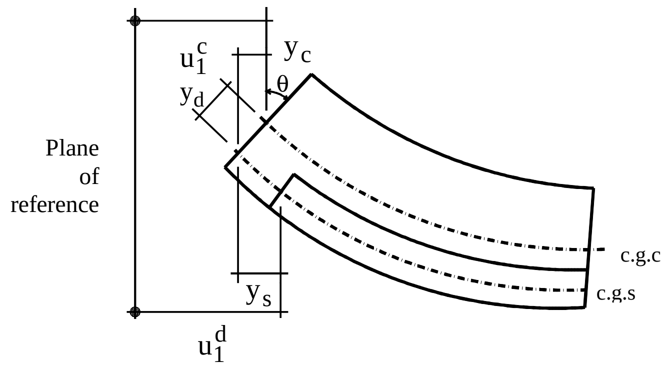


Figure 4-1. Schematic model of steel deck to concrete relative slip

where y_c = horizontal projection of y_d, y_d = depth of deck c.g. from concrete c.g., θ = rotation of cross sectional plane, y_s = horizontal slip of steel deck relative to the concrete, u_1^d = nodal displacement of steel deck beam element in d.o.f.-1 direction (horizontal), u_1^c = nodal displacement of concrete beam in d.o.f.-1 direction (horizontal).

For end anchorages, spring elements are placed at the supports to produce resistance to horizontal movements of the concrete slab and steel deck relative to the support. The spring is attached to the bottom surface of the deck. A schematic diagram of the model is shown in Fig. 4-2.

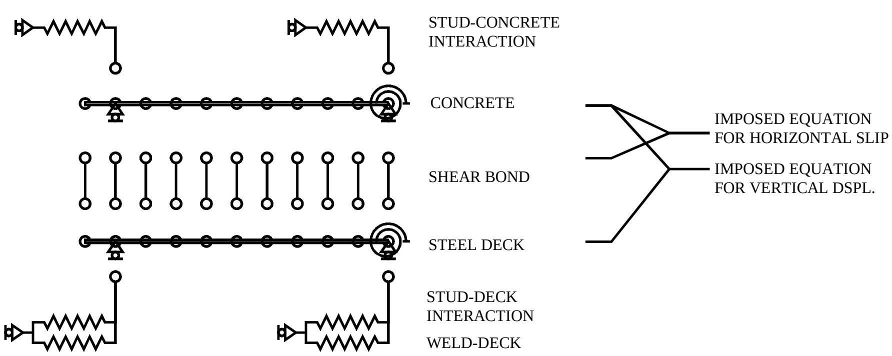


Figure 4-2. Typical finite element model

4.3.2. Material Model

Incremental plastic flow theory is applied for the steel and concrete materials whereas nonlinear elasticity theory is applied for end anchorages and shear bond interaction. J_2-plasticity (von Mises) with associative flow rule is used for the steel material of the steel deck. In this case, the yield surface is independent of the hydrostatic component of the stress vector as shown in Fig. 4-3. Although top flange buckling at the maximum positive moment region was observed in some specimens, no buckling is assumed in the model.

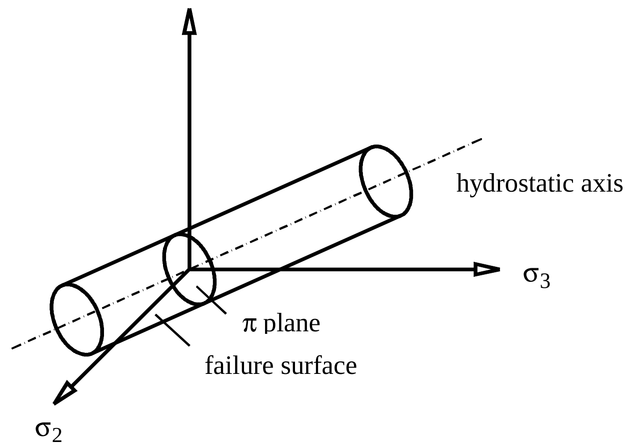


Figure 4-3. Von Mises yield surface in the principal stress space

The concrete material on the other hand, is pressure dependent. The general shape of failure surface for concrete material is illustrated in Fig. 4-4. ABAQUS uses the Drucker-Prager failure surface, a two-parameter model, for concrete material (Drucker and Prager 1952). This model is valid only for problems with low confining pressures (Hibbitt 1987). For a high confining pressure, many finer models of concrete failure surfaces are available, such as the Ottosen four parameter model (Ottosen 1977), Hsieh-Ting-Chen four parameter model (Hsieh et al. 1982), Willam-Warnke five parameter model (Chen 1982), etc. The Drucker-Prager model, however, is sufficient for one-way composite slabs. Moreover, because of the conical shape of the failure surface, singularity is only at the apex. Multi-vector return stress based on Koiter's (1953) approach is a common method to handle such singularity. Other methods such as a multiple single vector return (Widjaja 1997b) may improve the accuracy of the former method.

Recent developments in the application of fracture mechanics to concrete, in particular, concrete in tension, enabled a fracture mechanics model to be used for the tensile portion of the concrete. This model can avoid the mesh sensitivity effect of a tension-stiffening model. Further, in this study, an associative flow and isotropic work hardening rule is assumed.

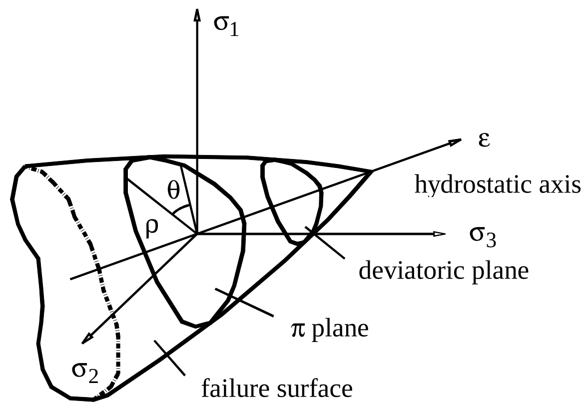


Figure 4-4. Concrete failure surface in principal stress space

The uniaxial stress-strain relation for concrete in compression is modeled using the Saenz equation up to the peak value (Saenz 1964). This model has been successfully used by Razaqpur and Nofal (1990) to model a composite bridge. The expression of Saenz equation is given by:

$$\sigma = \frac{E_o \varepsilon}{1 + \left(\frac{E_o}{E_{sc}} - 2\right)\frac{\varepsilon}{\varepsilon_{cu}} + \left(\frac{\varepsilon}{\varepsilon_{cu}}\right)^2} \tag{4-3}$$

where σ and ε are the stress and the corresponding strain of the concrete respectively, E_o and E_{sc} are the initial and the secant modulus of elasticity, respectively, ε_{cu} = concrete strain at the peak compressive stress. The descending branch of concrete-stress-strain relation is omitted in this beam model configuration to preserve stability of the system when compressive strength of concrete is approached. Figure 4-5 shows the concrete stress strain relation.

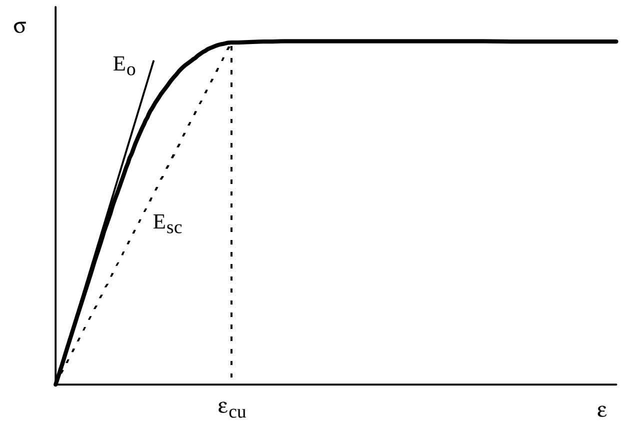


Figure 4-5. Concrete uniaxial compressive stress-strain relation

The backward Euler integration scheme is used in the plastic analysis. The scheme assumes that the return of the stress state to the yield surface is normal to the final yield surface (note that the yield surface keeps changing to follow the work hardening rule when plastic flow occurs).

Finally, a nonlinear elastic model is used to model end anchorages (welds or shear studs) and shear bond interaction. The force-displacement relation of these end anchorages and shear bond interactions were obtained from elemental tests as presented in Chapter 2. Typical shear bond interaction is shown in Fig. 4-6 and typical shear stud to steel deck and puddle weld to steel deck interactions, respectively, are shown in Figs. 4-7(a) and (b).

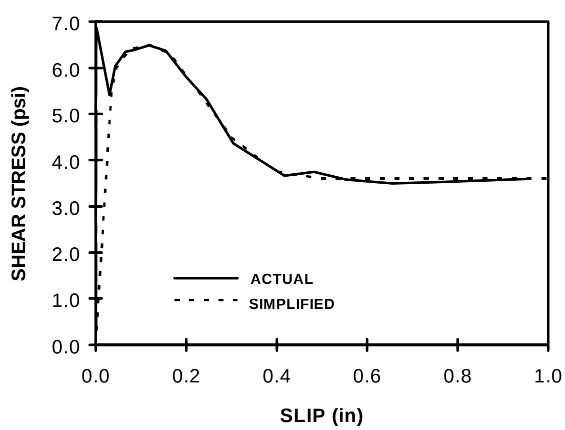
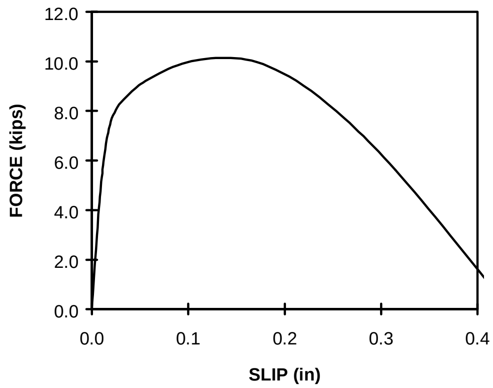
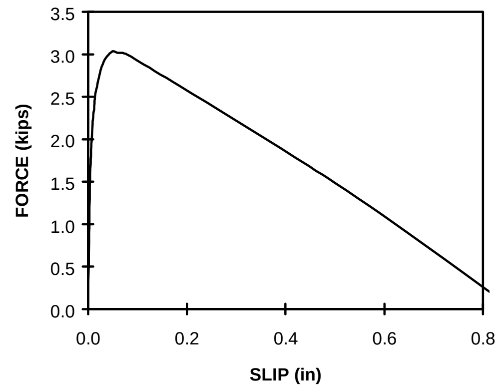


Figure 4-6. Typical shear bond shear stress vs. slip



(a)



(b)

Figure 4-7. (a) Shear stud to steel deck interaction, and (b) puddle weld to steel deck interaction

4.4. Method of Analysis

Among the three sources of nonlinearity: material, geometrical and boundary, only the first two are applicable to composite slab problems in this study. Both the material and geometrical nonlinearity were applied in the analyses. The integration scheme used to trace the equilibrium path was the arclength method with a cylindrical constraint surface as suggested by Crisfield (1981). The cylindrical constraint surface converges much faster than the general spherical one. Despite the problem with inconsistency in the physical units used in its constraint equations (Yang and McGuire 1985; Chen and Blandford 1993), no serious problem related to this inconsistency was reported. However, Widjaja (1997a) shows that the method is sensitive to the selection of physical units used. A choice of units that make the order of magnitude of each d.o.f. type (rotation, translational, etc.) comparable may improve the performance of the method. Other problems were indicated by Carrera (1994), such as no convergence due to a relatively large load step, very slow or no convergence due to oscillation near the equilibrium path, or no real root that satisfies the constraint surface. These later problems can be overcome by avoiding the use of large step sizes. Figure 4-8 illustrates the method with a general constraint surface.

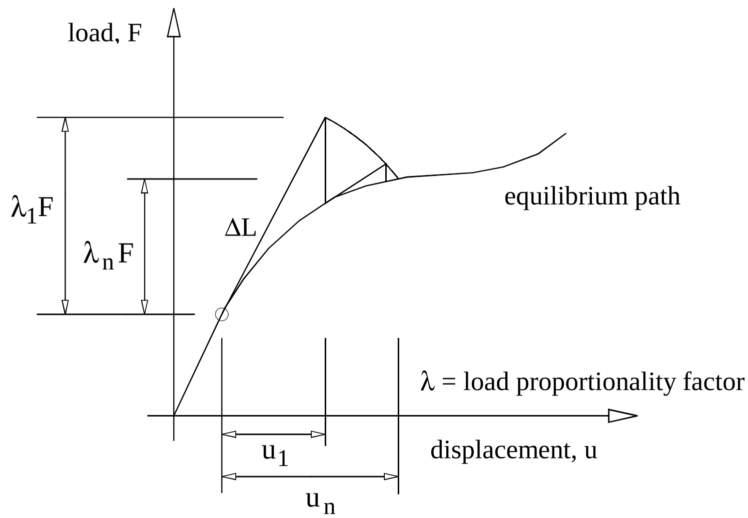


Figure 4-8. General arclength method

4.5. Results of Analysis and Discussion

Finite element analyses have been performed to simulate composite slab tests and results are listed in Table 4-1. Parameters of each slab specimens are listed in Table 3-1. Among the analysis results, load vs. mid-span deflection and load vs. end-slip response histories of slab-4 (studded slab with trapezoidal deck profile), slab-15 (studded slab with re-entrant deck profile) and slab-21 (welded slab, shored during the construction) are shown in Figs. 4-9, 4-10 and 4-11, respectively.

Table 4-1. Ultimate slab capacity: finite element vs. test results

SLAB #	FEM psf	TEST psf	RATIO TEST/ FEM	SLAB #	FEM psf	TEST psf	RATIO TEST/ FEM
1	627	730	1.16	15	985	1017	1.03
2	617	700	1.13	16	1037	1185	1.14
3	577	600	1.04	17	506	565	1.12
4	543	600	1.10	18	264	368	1.40
5	480	490	1.02	19	537	523	0.97
6	565	590	1.04	20	496	523	1.05
7	293	375	1.28	21	456	467	1.03
8	480	490	1.02	22	441	494	1.12
9	775	900	1.16	23	408	507	1.24
10	790	900	1.14	24	534	621	1.16
11	733	750	1.02	25	534	559	1.05
12	799	870	1.09	26	353	498	1.41
13	409	480	1.17	27	353	455	1.29
14	364	500	1.37				

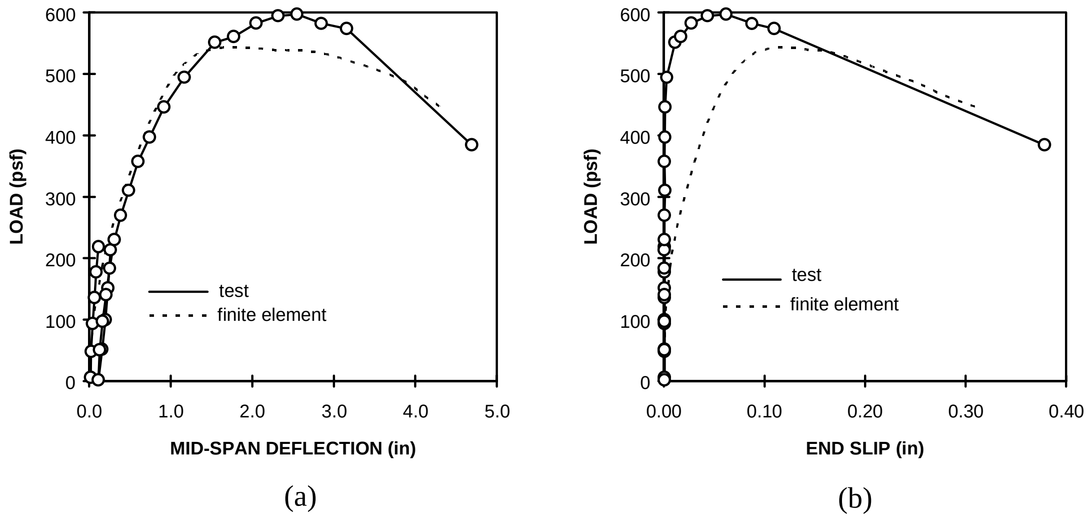


Figure 4-9. Slab-4: (a) Load vs. mid-span deflection. (b) Load vs. end-slip

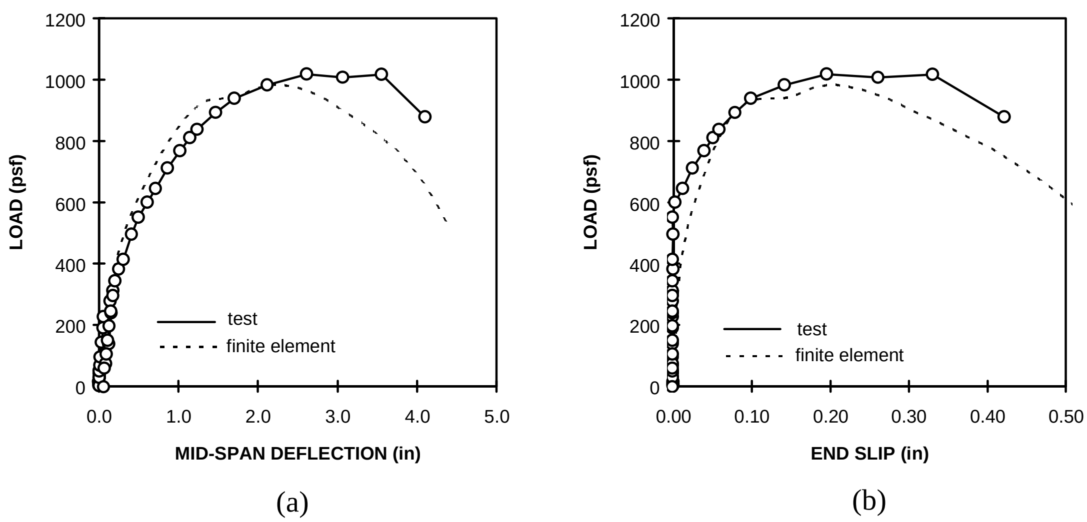
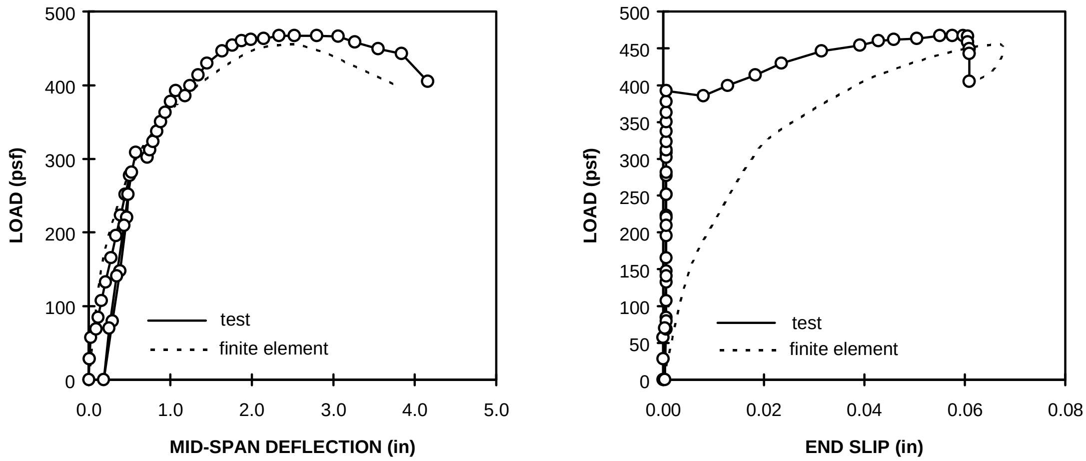


Figure 4-10. Slab-15: (a) Load vs. mid-span deflection. (b) Load vs. end-slip



(a) (b)

Figure 4-11. Slab-21: (a) Load vs. mid-span deflection. (b) Load vs. end-slip

Figure 4-12 shows graphical comparison of predicted vs. test values of slab strength. It can be seen from the figure, the predicted values for studded slabs fall within $\pm 15\%$ margin. For non-studded slabs, predicted values tend to be more conservative. This fact may be caused by the exclusion of clamping force to the steel deck and friction at steel deck-concrete interface at the supports.

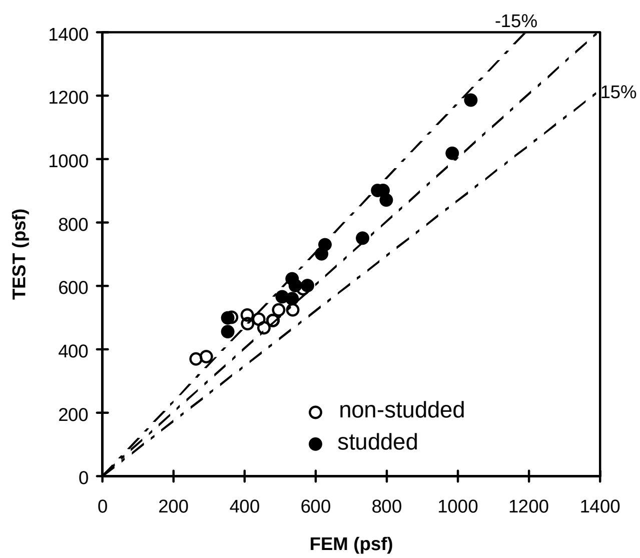


Figure 4-12. Composite slab strength: FEM vs. experimental

4.6. Concluding Remarks

Comparison of the finite element results to those of the tests for a relatively wide range of parameters demonstrates the ability of the method in predicting composite slab strength and behavior. This ability may reduce the number of expensive full-scale experimental tests for composite slabs. Further, the stress-strain response history of virtually any point in the system can be obtained.

In comparison to the iterative method, the nonlinear finite element method offers some advantages, such as the possibility to obtain stresses and strains at virtually any location of the slab. The method, however, requires more advanced user's knowledge than the iterative method. Therefore, the iterative method is more suitable for design purposes.