Computers and Concrete, Vol. 2, No. 5 (2005) 389-409
DOI: http://dx.doi.org/10.12989/cac.2005.2.5.389

Finite element modeling methodologies for FRP strengthened RC members

Sangdon Park† and Riyad Aboutaha‡

Department of Civil and Environmental Engineering, Syracuse University
255 Link Hall, Syracuse, NY 13244-1190, U.S.A.

(Received July 22, 2005, Accepted October 4, 2005)

Abstract: The Finite Element Analysis (FEA) is evidently a powerful tool for the analysis of structural concrete having nonlinearity and brittle failure properties. However, the result of FEA of structural concrete is sensitive to two modeling factors: the shear transfer coefficient (STC) for an open concrete crack and force convergence tolerance value (CONVTOL). Very limited work has been done to find the optimal FE Modeling (FEM) methodologies for structural concrete members strengthened with externally bonded FRP sheets. A total of 22 experimental deep beams with or without FRP flexure or/and shear strengthening systems are analyzed by nonlinear FEA using ANAYS program. For each experimental beams, an FE model with a total of 16 cases of modeling factor combinations are developed and analyzed to find the optimal FEM methodology. Two elements the SHELL63 and SOLID46 representing the material properties of FRP laminate are investigated and compared. The results of this research suggest that the optimal combination of modeling factor is STC of 0.25 and CONVTOL of 0.2. A SOLID 46 element representing the FRP strengthening system leads to better results than a SHELL 63 element does.

Keywords: CFRP composites; bridge piers; strengthening; finite element method.

1. Introduction

In the Finite Element Method approach, a three-dimensional finite element model is normally developed to examine the load carrying capacity, crack pattern, and failure mode of structural concrete. It has been found that the general structural behaviors obtained from the Finite Element Analysis show a close correspondence to the experimental test results. Previous researchers have also verified that the effect of an addition of an FRP sheet on the structural behavior of the FRP strengthened concrete beam can be analyzed by FEM with accuracy (Kim and Aboutaha). The 3-D finite element modeling is based on the dimensions and material properties of the experimental beams. The ANSYS program has a 3-D concrete solid element, SOLID65, which has crushing capability in compression and cracking capability in tension. The actual nonlinear material properties of concrete are accounted for. A LINK8, 3-D spar element, represents steel reinforcement. This element is capable of both tension and compression stresses, but not shear. It also replicates plastic deformation and creep. The SHELL63 and SOLID46 elements used to represent a layered FRP sheet are also modeled by following each component's actual nonlinear characteristics.

The Finite Element Analysis can predict the structural behavior of structural concrete with exact

† Ph. D., E-mail: sdpark@syr.edu
‡ Associate Professor, Corresponding Author, E-mail: rsabouta@syr.edu

certainty. However, finite element modeling of a reinforced concrete structure in a nonlinear analysis is time consuming work. Moreover, FEA of reinforced concrete either with or without FRP strengthening is sensitive to some modeling factors, i.e., the shear transfer coefficient, the convergence tolerance value, the sub step number, the number of elements, and the mesh size etc. Among those modeling factors, two values, i.e., the shear transfer coefficient (STC) for an open concrete cracks and the force convergence tolerance value (CONVTOL) have a critical impact on the result of FE Analysis. Both values also affect solution convergence. A total of 22 experimental beams with or without FRP flexural/shear strengthening systems are analyzed by nonlinear FEA using ANSYS program. For each experimental beam, an FE model with a total of 16 cases of modeling factor combinations, i.e., combination of 4 shear transfer coefficient values and 4 convergence tolerance values, were developed and executed to find the optimal FE modeling methodology. Both the SHELL63 and SOLID46, 3-D membrane element, are used to represent FRP laminate. The SHELL63, an Elastic Shell, is easy for modeling. The SOLID46 is a layered element of structural solid allowing up to 250 different layers with different fiber direction angles. The behavior of two membrane elements representing FRP sheet were compared.

2. FE modeling methodology

2.1. Element types and material properties

Three materials were involved in this study, i.e., concrete, steel, and FRP composites. A SOLID65, a LINK8, were used to represent the actual material properties of concrete, steel reinforcement, respectively. A SHELL63 or SOLID46 were used to represent the actual material properties of FRP laminate.

2.1.1. Concrete

A SOLID65, 3-D concrete solid element, simulates the structural behavior of concrete with or without reinforcement. This concrete element is similar to the other 3-D structural solid elements with the addition of special cracking and crushing capabilities (ANSYS manual). This element is capable of predicting the nonlinear behavior of concrete materials. The most important aspect of this solid element is the treatment of nonlinear material properties, such as the crushing capability in compression, the cracking capability in tension in three orthogonal directions, crushing, plastic deformation, and creep (ANSYS manual). The actual nonlinear material properties of concrete are accounted for. The stress strain response of concrete under uniaxial compression used in this FE analysis is the modified Hognestad parabola equation. The typical relationship is shown in Fig. 1. A multi-linear stress-strain relationship is used instead of the actual compressive nonlinear characteristics as shown in Fig. 2. A linearly elastic relationship with a modulus of elasticity *Eo* up to one third of the ultimate strain (ε_0) at the ultimate compressive strength of f'_c is used to represent material properties under uniaxial compression. The initial tangent modulus of elasticity, *Eo*, for reinforced concrete, is taken as $57000\sqrt{f'_c\,psi}$ (18.5 GPa). The uniaxial tensile strength used in this FE analysis is based on MacGregor as follows.

$$\sigma_t = 7.5\sqrt{f'_c}\ (psi)\ \text{or}\ 0.623\sqrt{f'_{cc}}\ (\text{MPa})$$

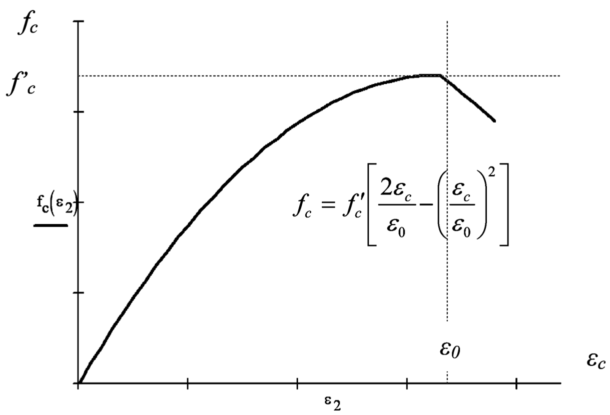


Fig. 1 Modified Hognestad parabola

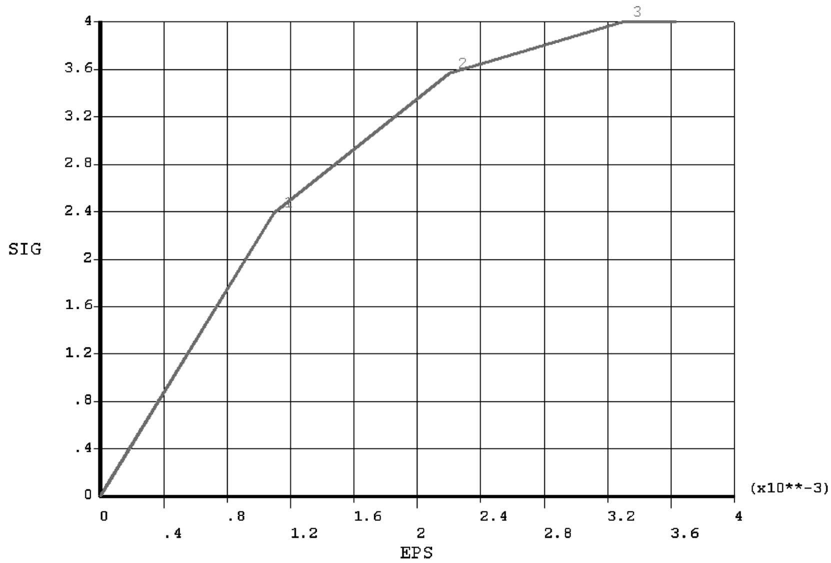


Fig. 2 Simplified stress-strain curve for 4 ksi concrete

Additional concrete material data used in the ANSYS data table are the shear transfer coefficients, tensile stresses, and compressive stresses, etc (ANSYS manual). Typical shear transfer coefficients range from 0.0 to 1.0, where 0.0 represents a smooth crack (complete loss of shear transfer) and 1.0 represents a rough crack (no loss of shear transfer) (ANSYS manual). These transfer coefficients may be applied for both the closed and open crack (ANSYS manual).

2.1.2. Steel reinforcements

A LINK8, 3-D spar element, is used to represent the steel reinforcement. The ANSYS rebar element, LINK8, is capable of handling tension and compression stresses, but not shear. It is also capable of plastic deformation and creep (ANSYS manual). This element has two nodes having three degrees of freedom at each node. This is also capable of predicting the nonlinear behavior of

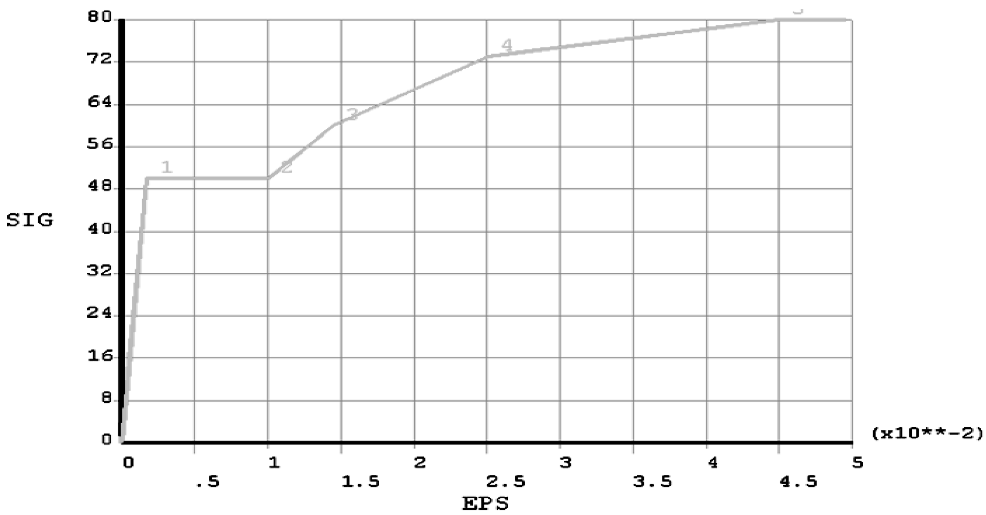


Fig. 3 Simplified stress-strain curve for grade 50 reinforcement

steel reinforcement. The actual nonlinear characteristic is simplified as a multi-linear stress-strain relationship in this study as shown in Fig. 3. The initial tangent modulus of elasticity, *Es*, for reinforcing bars are taken as 29E6 psi (200E3 MPa).

2.1.3. FRP laminates

Both the SHELL63 and SOLID46, 3-D membrane element, are used to represent FRP laminate. The elastic-perfectly plastic representation is assumed for the FRP in this study. The SHELL63, an Elastic Shell, has both bending and membrane capabilities. The element has four nodes having six degrees of freedom at each node: translations in the nodal *x, y,* and *z* directions and rotations about the nodal *x, y,* and *z*-axes (ANSYS manual). Stress stiffening and large deflection capabilities are included. Material data used to define the element are four nodes, four thicknesses, the elastic foundation stiffness, and the orthotropic material properties (ANSYS manual). The SOLID46, a 3-D 8-Node Layered Structural Solid element, is a layered version of the 8-node structural solid designed to model layered thick shells or solids (ANSYS manual). The element allows up to 250 different material layers with different material direction angles (ANSYS manual). The element has eight nodes having three degrees of freedom at each node: translations in the nodal *x, y*, and *z* directions (ANSYS manual). Material data used to define the element are eight nodes, layer thicknesses, layer material direction angles, and orthotropic material properties (ANSYS manual).

2.3. Nonlinear FE modeling

2.3.1. Shear transfer coefficients (STC)

A 3-D concrete model is meshed in such a way that both the steel reinforcements and FRP laminates can fit the FE nodes of the SOLID65 elements at the particular location. Then steel rebar and FRP are lumped and arranged in the SOLID model by following the test model as exactly as possible. In this study, the perfect bonding between materials, i.e., between concrete and steel rebar and between concrete and FRP laminate, was assumed. Poisson's ratios for concrete and steel used in this study are 0.2 and 0.3, respectively. The typical shear transfer coefficient for open crack face ranges from 0.0 to 1.0 (ANSYS manual). In this study, the shear transfer coefficients in ANSYS for closed cracks in the SOLID65 element are assumed to be between 0.9 and 1.0. A total of four coefficient values of shear transfer coefficients for open crack were used to find optimal coefficient data for ANSYS FE modeling. The shear transfer coefficients for open crack face used in this study were 0.1, 0.25, 0.5, and 0.85.

2.3.2. Convergence tolerance value (CONVTOL)

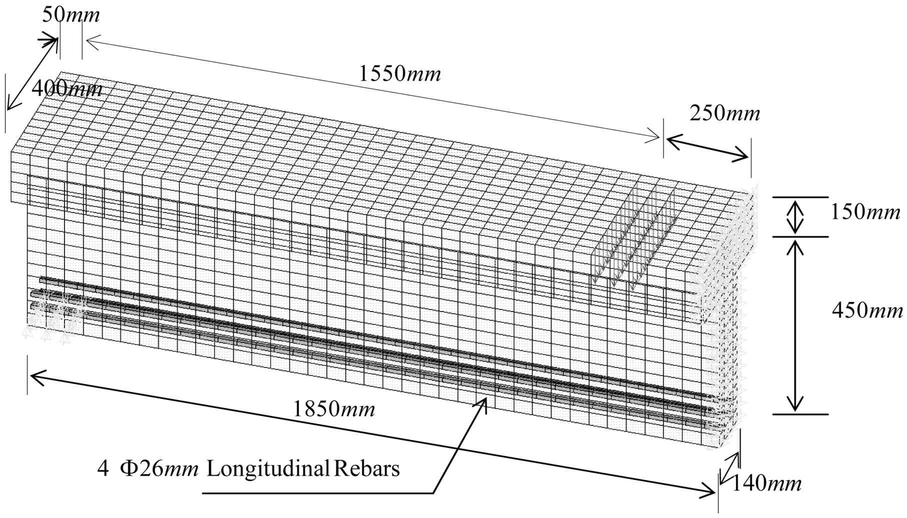
The program continues to do equilibrium iterations until the convergence criteria, CONVTOL, are satisfied (ANSYS manual). The default value of ANSYS force convergence tolerance is equal to 0.5%. And a moment convergence criterion is generally adequate. Usually the force convergence criterion controls the result and convergence in the FE analysis of reinforced concrete. By default, the program checks for force convergence by comparing the square root sum of the squares (SRSS) of the force imbalances against the product of VALUE TOLER (ANSYS manual). The default value of VALUE is the SRSS of the applied loads. The default value of TOLER is 0.005 if SOLCONTROL is on (default). Generally, using tighter convergence criteria improves the accuracy of results, but at the cost of more equilibrium iterations (ANSYS manual). Moreover, for brittle materials with large deflection such as structural concrete, a tight convergence value with smaller shear transfer coefficient causes convergence problem. In this study, the convergence tolerance values are changed by one or two orders of magnitude to get optimal value of CONVTOL. The CONVTOL used in this study are 0.005, 0.05, 0.1, and 0.2. A total of 22 experimental test beams with or without FRP flexural/shear strengthening systems were analyzed by nonlinear FEM approach using ANAYS. For each experimental test beam, a total of 16 cases of modeling factor combinations, i.e., four shear transfer coefficient values by four convergence tolerance values, were used to model each experimental test beam.

3. Nonlinear FEA of experimental beams

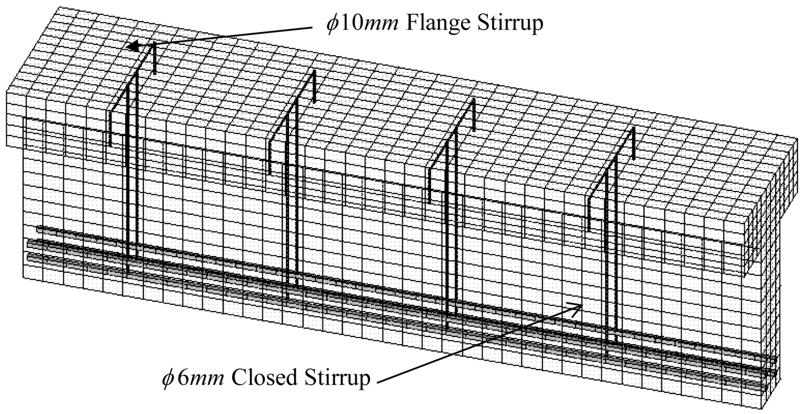
Several experimentally tested FRP strengthened concrete members were investigated and analyzed. These concrete members were tested by other researchers as indicated below.

3.1. Deniaud beam (Deniaud and Cheng 2001)

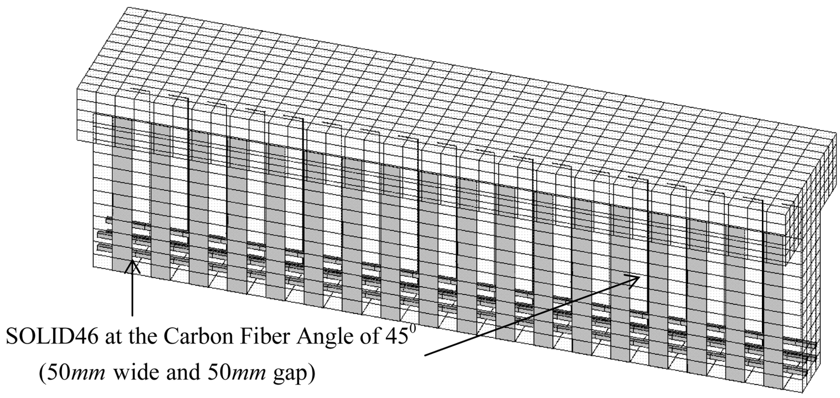
Only a longitudinal half of Deniaud beam was meshed and analyzed. Vertical and horizontal translations were restrained at the nodes of the left support and only horizontal translations were restrained at the nodes of the right hand side vertical surface as shown in Fig. 4 (a). The distance between support and the center of loading is 1550 mm, which is the same as test beam's, and whole model length is 1850 mm with height of 600 mm. A total of 3996 elements (SOLID65) for concrete and a total of 148 elements (LINK8) for bottom tension reinforcements are used to model Deniaud experimental beams. These numbers of elements for concrete and bottom reinforcement are same throughout the series of Deniaud beams. All FE models of Deniaud beams in this study have the same restraint conditions as shown in Fig. 4. A total of 6 FE models with 16 different modeling factors were developed and analyzed. The location and size of the FRP sheet were slightly modified



(a) FE modeling of T6NS beam

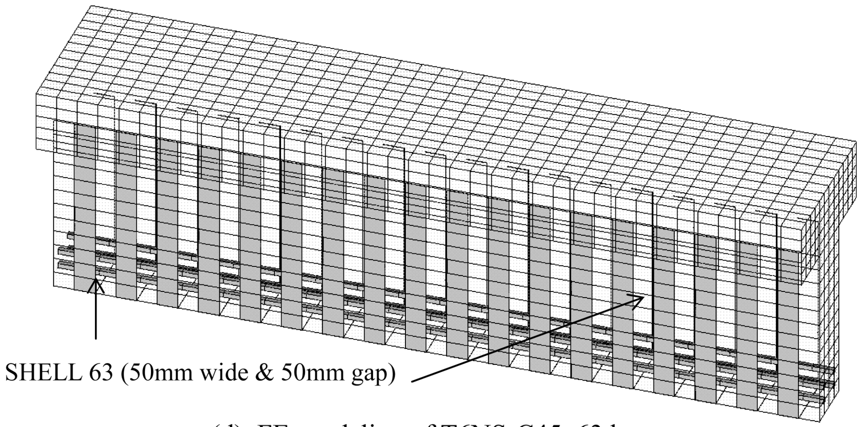


(b) FE modeling of T6S4 beam

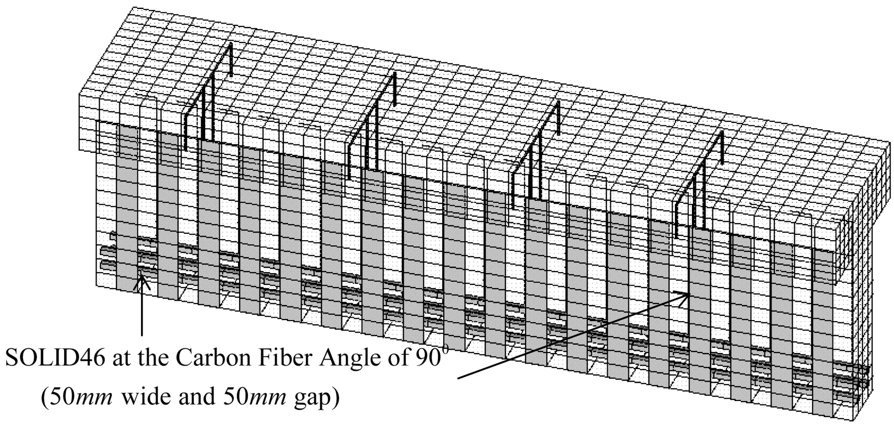


(c) FE modeling of T6NS-C45_46 beam

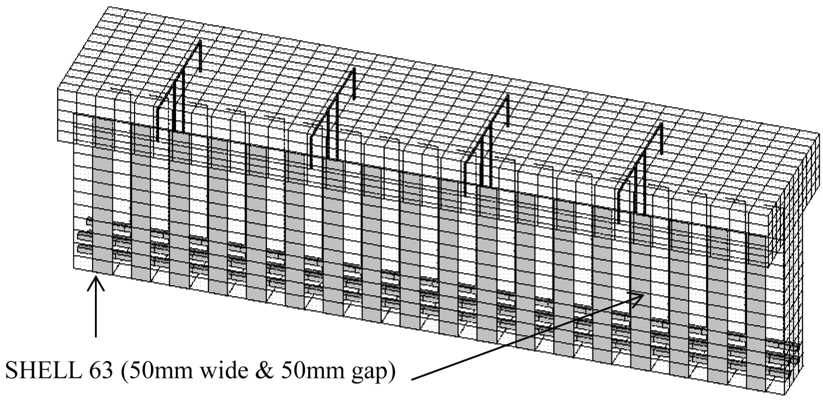
Fig. 4 FE modeling of Deniaud beam



(d) FE modeling of T6NS-C45_63 beam



(e) FE modeling of T6S4-C45_46 beam



(f) FE modeling of T6S4-C45_63 beam

Fig. 4 Continued

to follow concrete mesh size, but the area of FRP wasn't changed. Two types of ANSYS membrane elements representing FRP laminate (SHELL63 and SOLID46) were used to study the structural

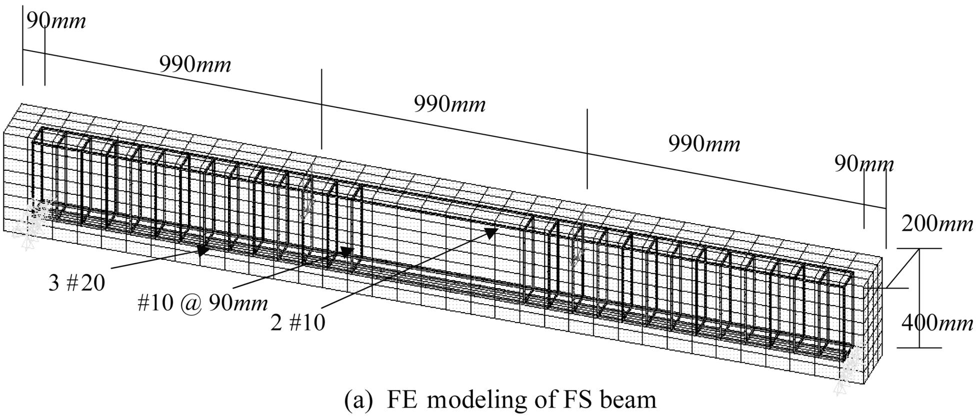
Table 1 The number of elements for Chaallal beams

Test beam	Material	FE element	Number of element
FS	Concrete	SOLID 65	1280
	Steel Bars	LINK8	453
RS	Concrete	SOLID 65	1280
	Steel Bars	LINK8	277
	FRP	SHELL63	300

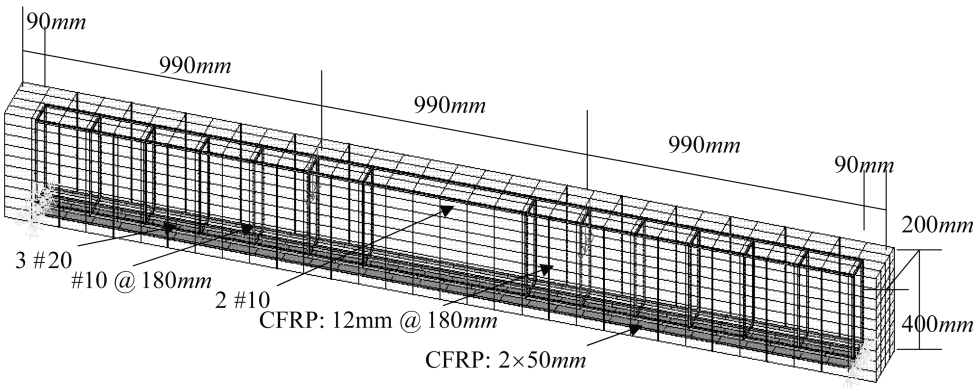
behavior of CFRP strengthened beams (T6NS-C45 and T6S4-C90).

3.2. Chaallal beams (Chaallal, Nollet and Saleh 1998)

Full size Chaallal beams were modeled and analyzed. The horizontal distance between the support and loading of the FE model is 990 mm (test beam: 1000 mm), and the whole model length is 3150mm with a height of 400 mm and a width of 200 mm. The number of elements is summarized in Table 1. All FE models of Chaallal beams in this study have four point loading systems as shown



(a) FE modeling of FS beam



(b) FE modeling of RS beam

Fig. 5 FE modeling of Chaallal beams

in Fig. 5. A total of 2 FE models with 16 different modeling factors were developed and analyzed. The beam length is slightly modified so that the location of steel stirrups and FRP sheet can fit with the meshing, but the area of FRP wasn't changed. The ANSYS membrane element SHELL63 was used to represent FRP sheet.

3.3. Fanning beams (Fanning and Kelly 1999)

Full size Fanning beams were modeled and analyzed. The horizontal distance between support and loading of the FE model is 500 mm and the whole model length is 1550 mm with a height of 240 mm and width of 150 mm. The number of elements is summarized in Table 2. All FE models of Fanning beams in this study have three point loading systems as shown in Fig. 6. A total of 6 FE models with 16 different modeling factors were developed and analyzed. The dimensions of the FE model are same as those of experimental beams. Minor modifications of the location of steel stirrups and FRP sheet were made to fit with the concrete meshing size, but the area of FRP wasn't changed. The ANSYS membrane element SHELL63 was used to represent FRP sheet.

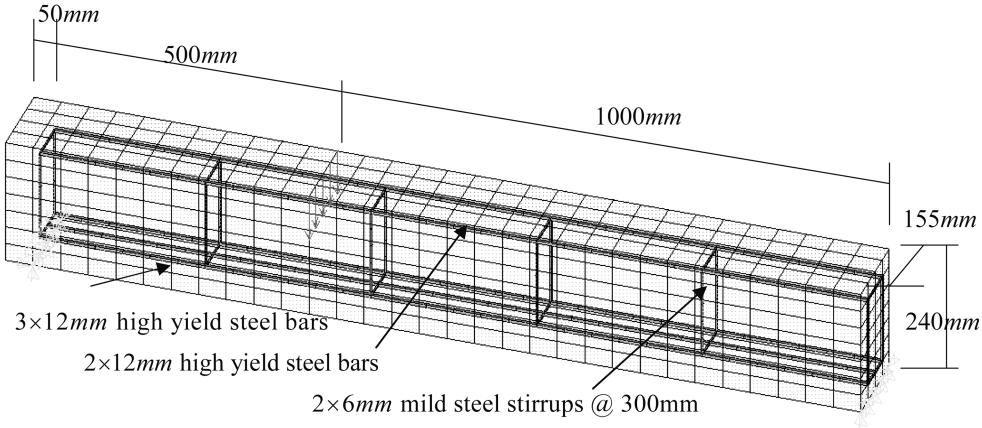
3.4. FEA results of experimental beams

3.4.1. Optimal FE modeling factor

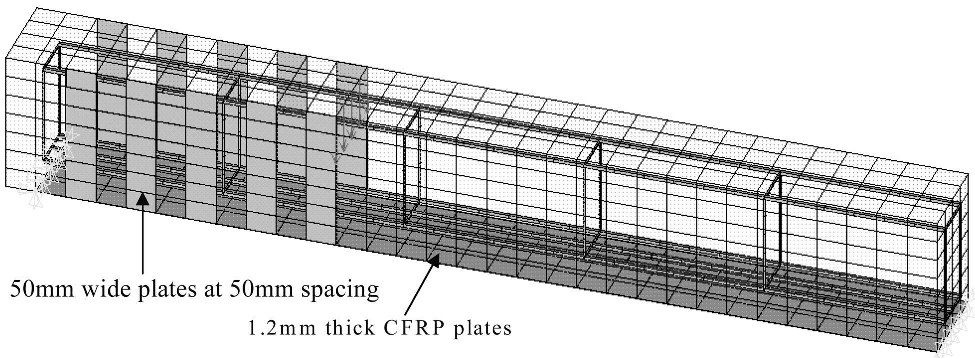
Fig. 7 shows the summary of descriptive statistics for a total of 352 FEA results, where the result 68 kN of SN-2 beam is excluded in static analysis since it is too small comparing to others. The

Table 2 The number of elements for Fanning beams

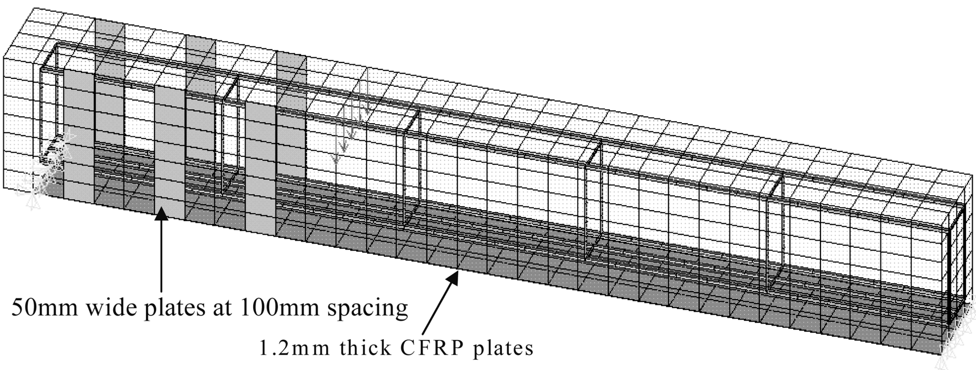
Test beam	Material	FE element	Number of element
Reference beam	Concrete	SOLID 65	868
	Steel bars	LINK8	234
FS-1 and FS-2	Concrete	SOLID 65	868
	Steel bars	LINK8	234
	FRP	SHELL63	198
FS-2	Concrete	SOLID 65	868
	Steel bars	LINK8	234
	FRP	SHELL63	162
SP-1	Concrete	SOLID 65	868
	Steel bars	LINK8	234
	FRP	SHELL63	112
SP-2	Concrete	SOLID 65	868
	Steel bars	LINK8	234
	FRP	SHELL63	84
SP-3	Concrete	SOLID 65	868
	Steel bars	LINK8	234
	FRP	SHELL63	56



(a) FE modeling of Reference beam



(b) FE modeling of FS-12 beam

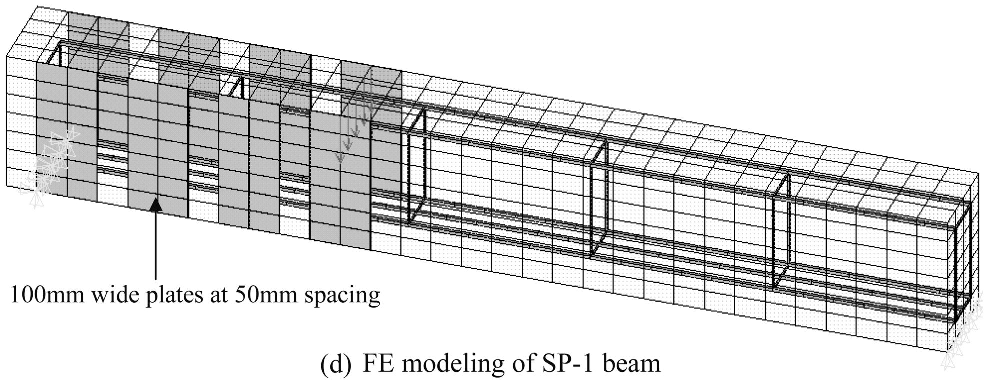


(c) FE modeling of FS-3 beam

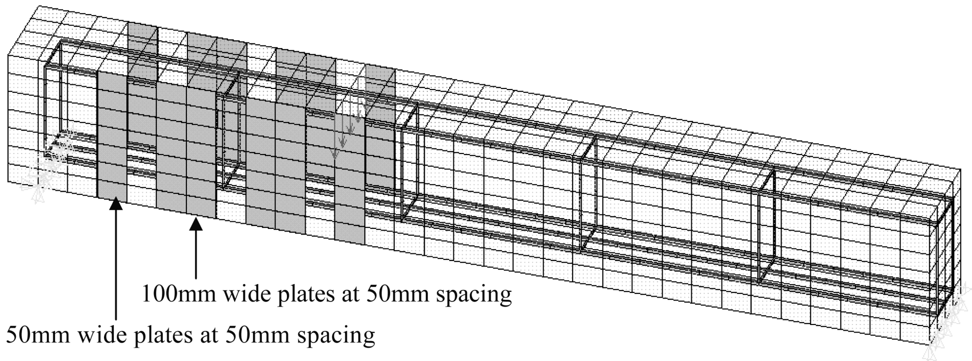
Fig. 6 FE modeling of Fanning beams

whole FEA result data has a variance of 395.885, which shows how wide the divergence of the data is. Differences in the Figure and Table are calculated by following equation:

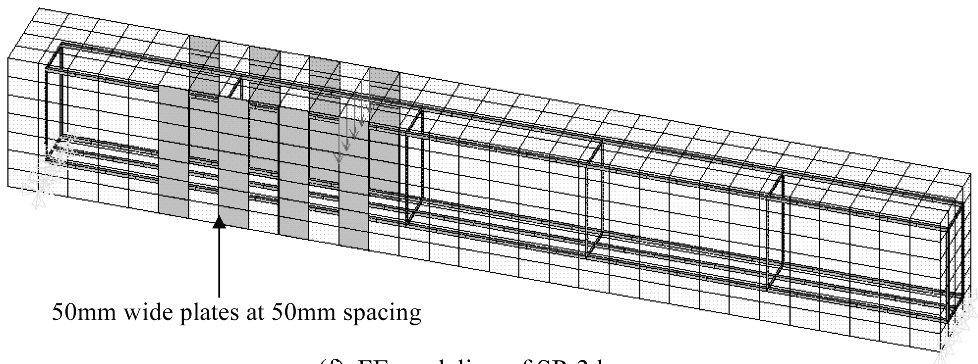
$$Difference = \frac{Test\ \ Result - STM\ \ Ressult}{Test\ \ Result} \times 100(\%)$$



(d) FE modeling of SP-1 beam



(e) FE modeling of SP-2 beam



(f) FE modeling of SP-3 beam

Fig. 6 Continued

Fig. 8 illustrate scattered individual plots of differences with the median value by variables of shear transfer coefficient and the convergence tolerance value for open crack, respectively. These plots demonstrate that the ultimate load capacity increase as STC and CONVTOL increase, where a negative value of difference means that the FEA result exceeds the test results by such a difference (%). Both STC for an open concrete cracks and CONVTOL give critical effect on the result of the FE Analysis of reinforced concrete. Fig. 9 depicts the histogram of difference for individual modeling factors and Table 3 summarizes the descriptive statistics of the difference for

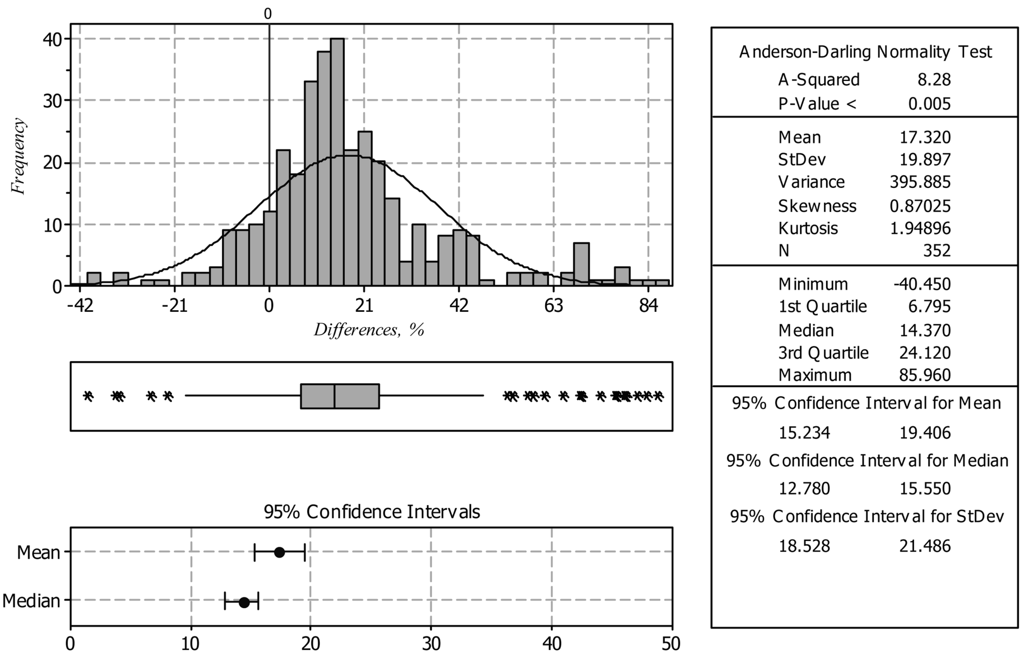


Fig. 7 Summary of descriptive statistics for 352 FEA results

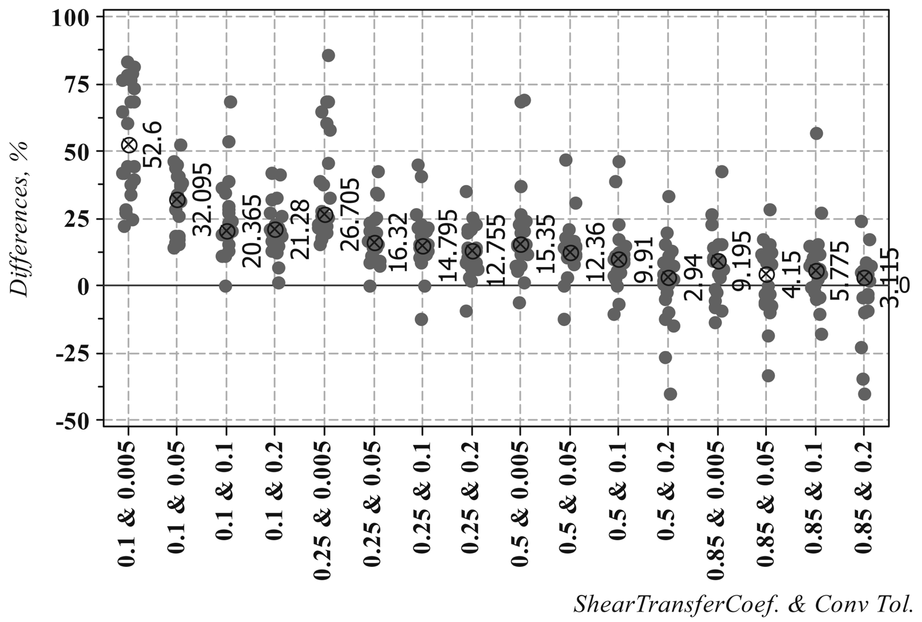


Fig. 8 Individual value plots of differences with median value

each modeling factor. An inspection of these figure and table indicates that the test results are best represented by FEA results with the modeling factor of “0.25 and 0.2”, which has the smallest standard deviation of 9.17 with a mean difference of 12.824%, but has one negative

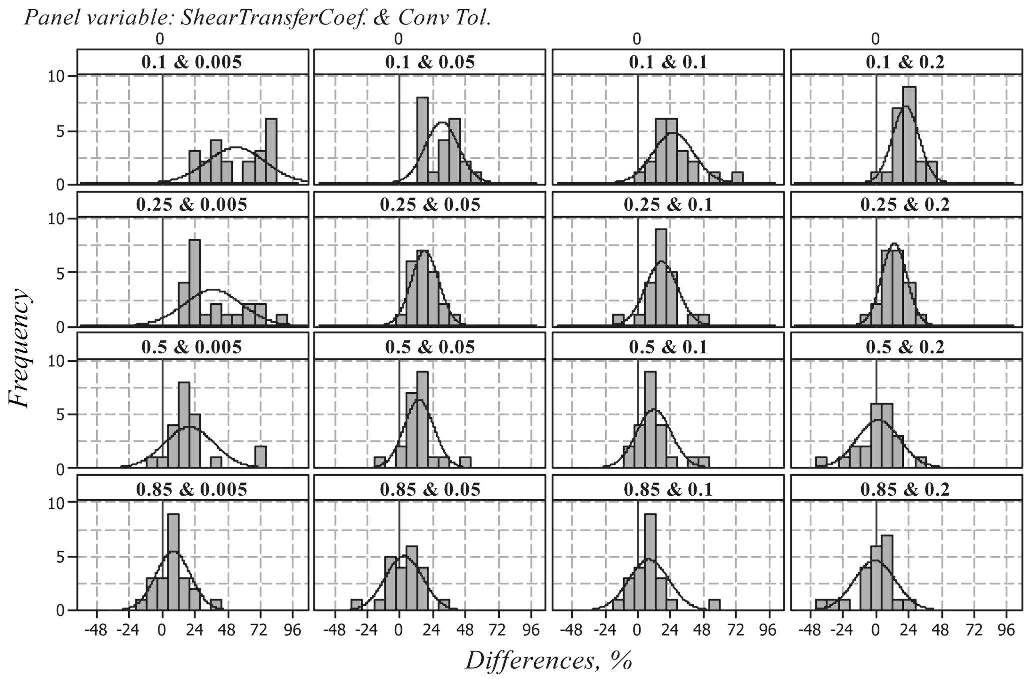


Fig. 9 Histogram of difference %

Table 3 Summary of descriptive statistics in difference %

STC & CONVTOL	Minimum	Mean	Maximum	Median	StDev
0.1 & 0.005	22.19	54.45	83.61	52.60	21.19
0.1 & 0.05	14.10	30.29	52.71	32.10	12.25
0.1 & 0.1	0.00	25.06	68.40	20.37	14.83
0.1 & 0.2	1.52	21.29	41.72	21.28	9.79
0.25 & 0.005	15.55	37.40	85.96	26.71	20.94
0.25 & 0.05	0.00	17.86	42.82	16.32	10.03
0.25 & 0.1	-12.12	16.41	44.99	14.80	11.73
0.25 & 0.2	-9.09	12.82	35.12	12.76	9.17
0.5 & 0.005	-6.06	19.96	69.10	15.35	18.28
0.5 & 0.05	-12.12	13.45	47.21	12.36	11.01
0.5 & 0.1	-10.37	10.78	46.11	9.91	12.82
0.5 & 0.2	-40.45	1.23	33.29	2.94	15.65
0.85 & 0.005	-13.64	8.07	42.82	9.20	12.66
0.85 & 0.05	-33.33	2.35	28.52	4.15	13.79
0.85 & 0.1	-18.18	7.04	57.11	5.78	14.73
0.85 & 0.2	-40.45	-1.34	24.12	3.12	15.03

difference. For conservative analysis, the modeling factor of “0.25 and 0.05” is the best answer, providing the smallest mean, median, and standard deviation among those which don’t have negative difference. In conclusion, in order to get safe results, we could take the conservative

Table 4 FEA results of T6NS-C45_46 Test result: 427.5 kN

Cases	S.T.C	CONVTOL	Deflection	ULT. Strength	Differences(%)
1	0.1	0.005	3.96	114.00	73.31
2	0.1	0.05	12.80	294.00	31.18
3	0.1	0.1	13.41	306.00	28.37
4	0.1	0.2	15.69	336.00	21.35
5	0.25	0.005	13.54	330.00	22.75
6	0.25	0.05	14.76	354.00	17.13
7	0.25	0.1	20.65	390.00	8.71
8	0.25	0.2	22.44	420.00	1.69
9	0.5	0.005	15.95	396.00	7.30
10	0.5	0.05	18.25	414.00	3.09
11	0.5	0.11	17.27	426.00	0.28
12	0.5	0.2	53.37	600.00	-40.45
13	0.85	0.005	18.28	414.00	3.09
14	0.85	0.05	17.28	426.00	0.28
15	0.85	0.1	16.85	438.00	-2.53
16	0.85	0.2	50.06	600.00	-40.45

Table 5 FEA results of T6NS-C45_63 Test result: 427.5 kN

Cases	S.T.C	CONVTOL	Deflection	ULT. Strength	Differences(%)
1	0.1	0.005	1.80	70.00	83.61
2	0.1	0.05	10.54	240.00	43.82
3	0.1	0.1	4.39	135.00	68.40
4	0.1	0.2	10.50	250.00	41.48
5	0.25	0.005	1.40	60.00	85.96
6	0.25	0.05	11.39	280.00	34.46
7	0.25	0.1	8.75	235.00	44.99
8	0.25	0.2	14.00	330.00	22.75
9	0.5	0.005	12.40	315.00	26.26
10	0.5	0.05	11.52	295.00	30.95
11	0.5	0.11	9.66	260.00	39.14
12	0.5	0.2	10.88	285.00	33.29
13	0.85	0.005	11.45	315.00	26.26
14	0.85	0.05	13.42	360.00	15.73
15	0.85	0.1	16.31	410.00	4.03
16	0.85	0.2	16.44	405.00	5.20

modeling factor of STC = 0.25 and CONVTOL = 0.05 for FEA of structural concrete with or without FRP reinforcement. To get a close result, take the optimal modeling factor of STC = 0.25 and CONVTOL = 0.2.

Table 6 FEA results of T6S4-C45_46 Test result: 545.6 kN

Cases	S.T.C	CONVTOL	Deflection	ULT. Strength	Differences(%)
1	0.1	0.005	13.60	318.00	41.72
2	0.1	0.05	14.39	336.00	38.42
3	0.1	0.1	14.97	348.00	36.22
4	0.1	0.2	16.51	366.00	32.92
5	0.25	0.005	14.89	366.00	32.92
6	0.25	0.05	14.46	360.00	34.02
7	0.25	0.1	21.80	420.00	23.02
8	0.25	0.2	24.48	432.00	20.82
9	0.5	0.005	19.35	414.00	24.12
10	0.5	0.05	21.69	432.00	20.82
11	0.5	0.11	16.74	420.00	23.02
12	0.5	0.2	52.56	600.00	-9.97
13	0.85	0.005	18.73	420.00	23.02
14	0.85	0.05	23.66	450.00	17.52
15	0.85	0.1	19.04	462.00	15.32
16	0.85	0.2	49.81	600.00	-9.97

Table 7 FEA results of T6S4-C45_63 Test result: 545.6 kN

Cases	S.T.C	CONVTOL	Deflection	ULT. Strength	Differences(%)
1	0.1	0.005	3.13	102.00	81.30
2	0.1	0.05	10.99	258.00	52.71
3	0.1	0.1	10.55	252.00	53.81
4	0.1	0.2	14.24	318.00	41.72
5	0.25	0.005	8.74	228.00	58.21
6	0.25	0.05	12.47	312.00	42.82
7	0.25	0.1	13.05	324.00	40.62
8	0.25	0.2	14.36	354.00	35.12
9	0.5	0.005	13.24	342.00	37.32
10	0.5	0.05	10.66	288.00	47.21
11	0.5	0.11	10.78	294.00	46.11
12	0.5	0.2	19.16	438.00	19.72
13	0.85	0.005	11.27	312.00	42.82
14	0.85	0.05	14.79	390.00	28.52
15	0.85	0.1	8.23	234.00	57.11
16	0.85	0.2	15.72	414.00	24.12

3.4.2. Comparison of FE element SHELL63 and SOLID46 as FRP strips

Both SOLID46 and SHELL63, 3-D membrane elements, were used to represent FRP laminate. T6NS-C45_46 FE model and T64S-C90_46 FE model used the SOLID 46 layered element to represent CFRP strips at the carbon fiber angle of 45 and 90 degrees, respectively. To model FRP sheet, an additional four nodes are needed to create the SOLID46 element. This is time consuming work compared to modeling the SHELL63 element. T6NS-C45_63 FE model and T64S-C90_63 FE model used SHELL63 membrane element to represent CFRP strips. FRP elements are attached to the concrete elements by sharing four nodes of SHELL63 with four nodes of SOLID65 without creating any more nodes. In a comparison of FEA results, the analysis results of FE Models with SOLID46 are close to the test results than those of FE Models with SHELL63 as shown in Table 4 to Table 7. To get more exact results, the SOLID46 element is recommended for FRP strips. Statistics of FEA using SOLID46 have a mean difference of 15.595 with a standard deviation of 22.503. Those of FEA used SHELL63 have a mean difference of 41.112 with standard deviation of 19.974. SOLID46 element is recommended to represent CFRP laminate.

4. Analysis of CFRP strengthened concrete bridge pier

There are three categories related to strengthening or retrofitting deteriorated RC bridges. These are flexural strengthening, shear strengthening, and confinement. The flexural and shear retrofit scheme of bonding FRP composites to the structurally deficient bridge pier is shown in Fig. 10. Fig. 10 (a) shows cracked pier cap beam. Fig. 10 (b) illustrates flexural and shear retrofit schemes using FRP strips. The bonded FRP composites serving as additional reinforcement can increase stiffness and load carrying capacity, and also provide crack control for the concrete members. The concrete pier is analyzed by the proposed FE modeling methodology for FRP reinforced deep concrete members.

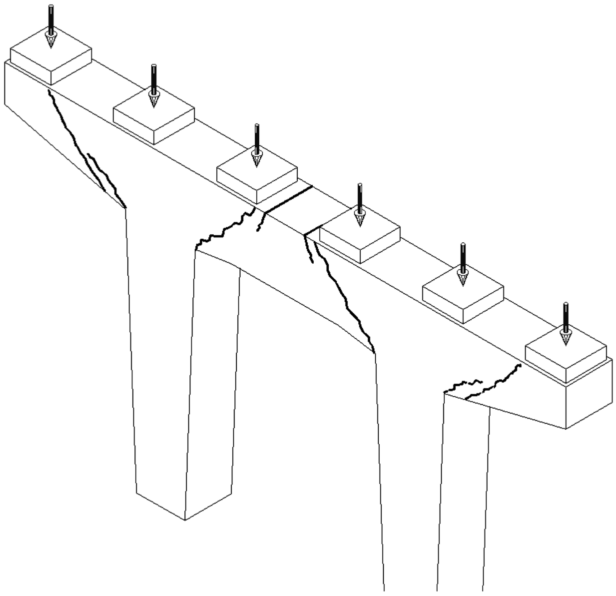
The column considered in this study has a slenderness ratio of 13.3, which is smaller than that of the limiting slenderness of 22. The column of the pier is a stocky column, therefore, slenderness effect does not need to be considered. Short or stocky columns fail by material failure and the nominal column strength of this pier is 12,696 kips. Therefore the column will not fail until the cap beam fails. In this study it is assumed that the columns do not affect the behavior of the upper structure of the pier. The left half of pier cap is considered. Hinge and roller boundary conditions are imposed in the column and cap beam, respectively. Material properties used in this FE Analysis are summarized in Table 8. The dimension and details of the pier cap beam are illustrated in Fig. 11. The shear transfer coefficient for open concrete crack and the convergence tolerance value for force used in this analysis are 0.25 and 0.2, respectively. The material properties of SOLID46 element used to represent CFRP strips is replicated by following the component's nonlinear characteristics.

4.1. Original pier cap beam

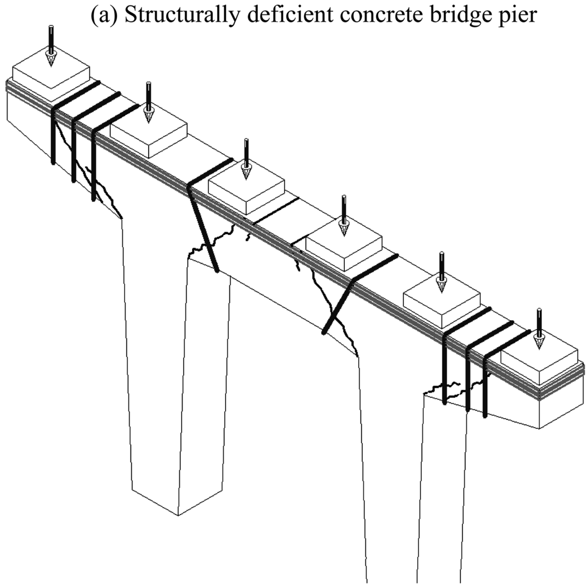
By taking advantage of the symmetry of the pier cap beams, a only half of the upper part of the pier cap beam is modeled with proper boundary conditions. The concrete is meshed following steel reinforcement lay out. The modeling of steel bar follows the steel reinforcement of the existing bridge pier model as shown in Fig. 12. FE element types and the number of elements are summarized in Table 9.

4.2. FRP strengthened pier cap beam

The cracked cap beam is strengthened by bonding FRP strips to the cap beam surface. The



(a) Structurally deficient concrete bridge pier



(b) FRP strengthened concrete bridge pier cap beam

Fig. 10 Flexural and shear retrofit schemes

Table 8 Material properties for pier cap beam

Material	Elastic modulus	Compressive strength	Tensile strength
Concrete	3650 ksi	4 ksi	0.475 ksi
Steel Bar	29000 ksi	50 ksi	50 ksi
FRP	22500 ksi	NA	270 ksi

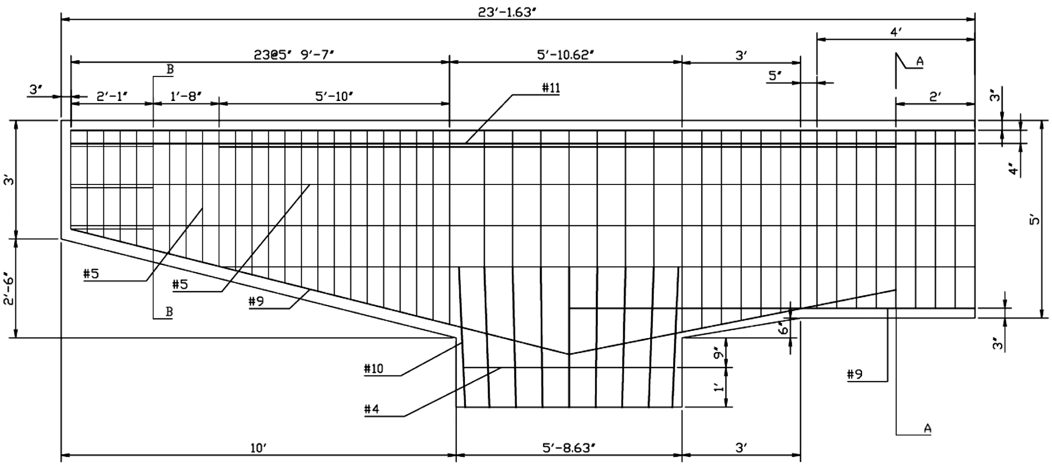


Fig. 11 Details of pier cap beam

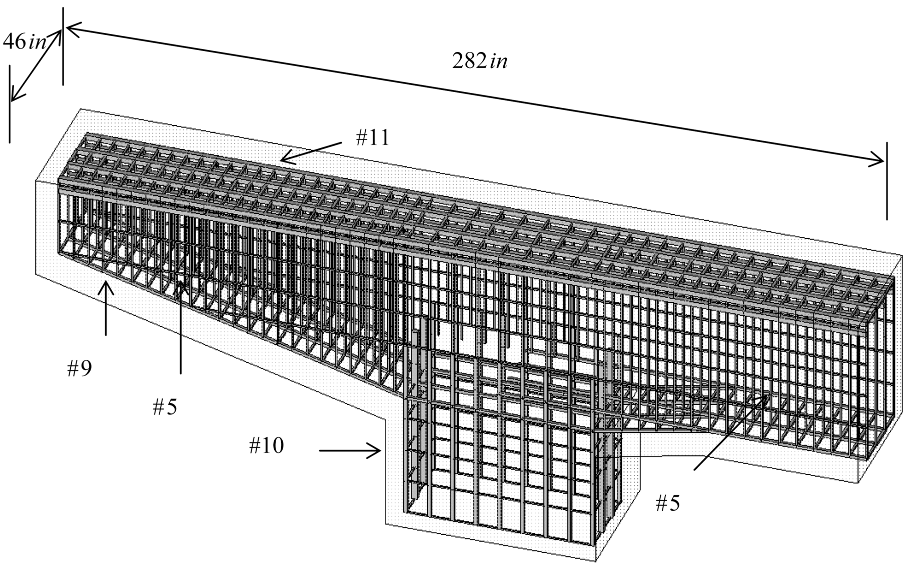


Fig. 12 FE modeling of pier cap beam

Table 9 Element types and number of elements

Cap beam	Material	FE element	Number of element
Original beam	Concrete	SOLID 65	2160
	Steel bars	LINK8	2291
FRP Strengthened beam	Concrete	SOLID 65	2160
	Steel bars	LINK8	2291
	FRP	SOLID46	256
		45 degree fiber direction	

amount of FRP strips and the retrofit scheme are illustrated in Fig. 13. The FE model of FRP strengthened capbeam is shown in Fig. 14.

4.3. FEA results of pier cap beam

Fig. 15 shows the analysis results of structural concrete with or without FRP strengthening using the proposed FEM approaches. The result of FEM is compared with the results of Strut-and-Tie Mothod (STM) approach proposed by Park and Aboutaha (2005). The desired load capacity of FRP strengthening, 945 kips, is 1.25 times of original beam strength, 756 kips. However, the desired strength can't be reached due to early failure of concrete crushing before FRP rupture as shown in Fig. 16. The STM result also shows the same failure mode. The table in the Fig. 15 summarizes the

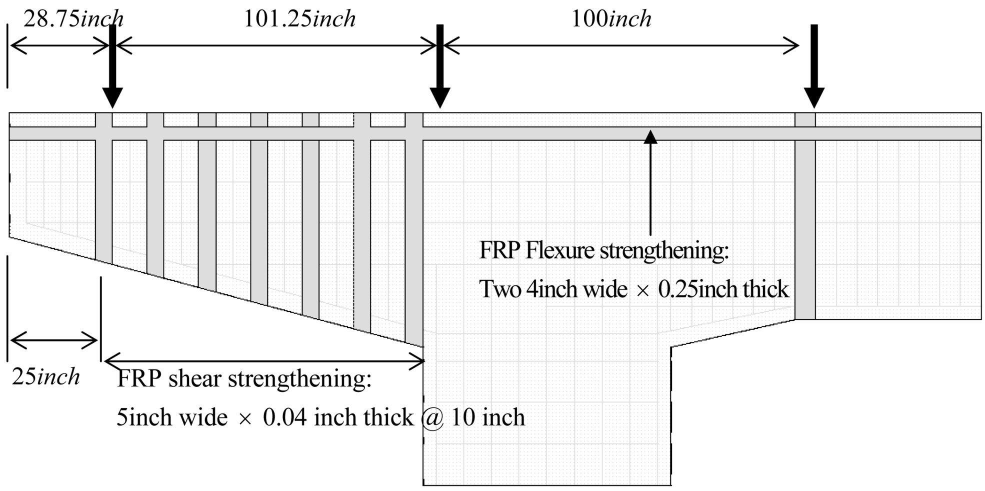


Fig. 13 FRP retrofit scheme

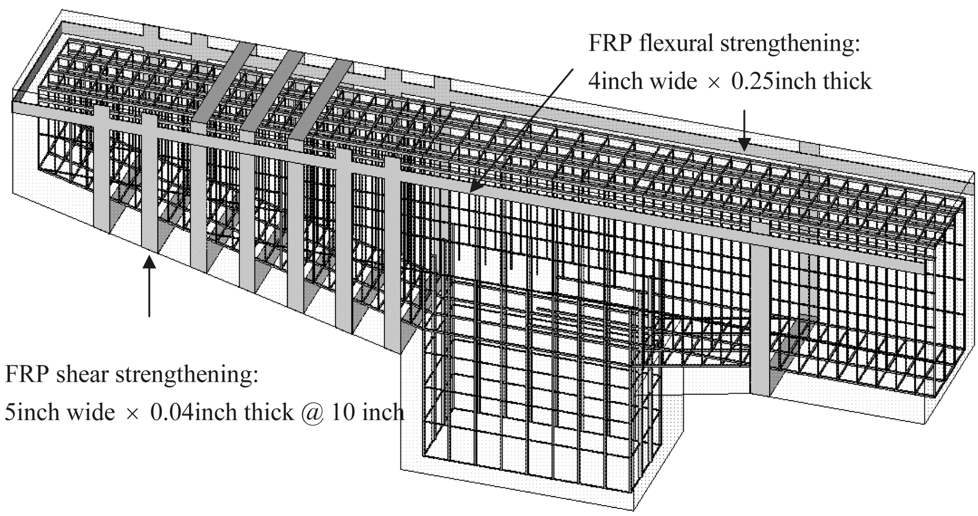


Fig. 14 FRP retrofit scheme and FE modeling

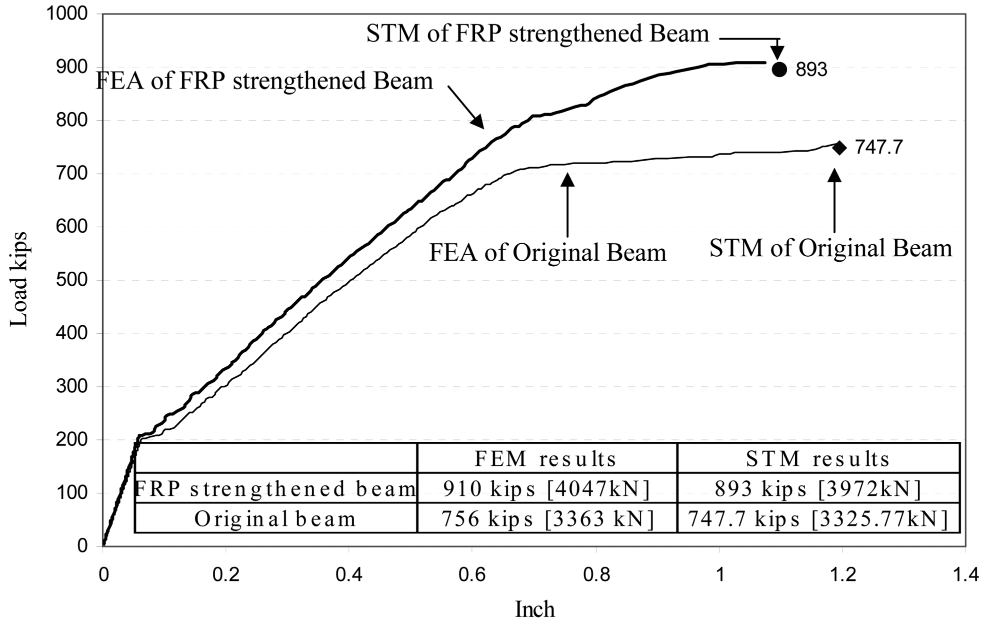


Fig. 15 Results of proposed FEM approach for RC cap beam

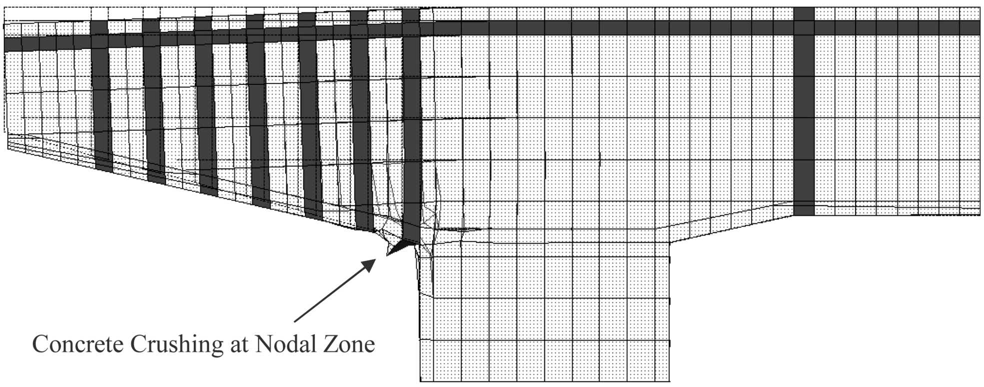


Fig. 16 Deformed shape of FRP retrofit cap beam at the failure load of 915 kips

FEM result and STM result for cap beams with and without FRP strips. For FRP strengthened cap beam, FEM results is 910 kips and STM results is 893kips. For original cap beam, FEM result is 756 kips and STM result is 747.7 kips. Two approaches are well correlated each other.

5. Conclusions

In the Finite Element Method approach, a three-dimensional Finite Element Model was developed to examine the load carrying capacity, crack pattern, and failure mode of structural concrete. A total of 22 experimental beams with or without FRP shear strengthening were analyzed by nonlinear FEM using ANSYS. For each experimental beam, the FE model with a total of 16 cases of modeling factor combinations, i.e., 4 shear transfer coefficient values by 4 convergence tolerance values were developed and analyzed to find the optimal FE modeling methodology. Both SHELL63 and SOLID46, 3-D membrane element, were used to represent FRP laminate. The behavior of two FRP materials was compared. The left half of the capbeam of concrete bridge pier was analyzed by the proposed FE modeling methodology for FRP strengthened deep concrete members. A SOLID46 element was used to represent FRP strips.

Based on the analysis of experimental test members, the following conclusions could be drawn:

1) The FE Method is evidently a powerful tool for the structural analysis of reinforced concrete members in the inelastic range and having brittle failure properties.

2) FEA of reinforced concrete either with or without FRP strengthening is sensitive to two modeling factors, i.e., the shear transfer coefficient (STC) for open crack and the force convergence tolerance value (CONVTOL).

3) The ultimate load capacity calculated by FEM increases as STC and CONVTOL increase.

4) For conservative design, the conservative modeling factor of STC = 0.25 and CONVTOL = 0.05 for FEA of structural concrete with or without FRP reinforcement are recommended.

5) For conservative and least conservative design, the optimal modeling factor of STC = 0.25 and CONVTOL = 0.2 are recommended.

6) The analysis results of FE Models with SOLID46 are closer to the test results than those of FE Models with SHELL63. To achieve better results, SOLID46 element is recommended for FRP strips.

References

Aboutaha, R. S. (2002), "Ductility of CFRP strengthened concrete bridge girders", Department of Civil and Environmental Engineering, Syracuse University, Syracuse, NY.

ANSYS release 5.7. *ANSYS Theory Manual,* 12th Edition, SAS IP Inc.

Buyukozturk, O., Hearing, B. and Gunes, O. "FRP strengthening and repair: where do we go from here?", Messachusetts Institute of Technology, Department of Civil Environmental Engineering, Cambridge, MA.

Chaallal, O., Nollet, M. J. and Saleh, K. (1998), "Use of CFRP strips for flexure and shear strengthening of RC members", *2nd Conference on Composites in Infrastructure*, June.

Deniaud, C. and Roger Cheng, J. J. (2001), "Shear behavior of reinforced concrete T-beams with externally bonded fiber-reinforced polymer sheets", *ACI Struct. J.*, May-June, 386-394.

Fanning, P. and Kelly, O. (1999), "Shear strengthening of reinforced concrete beams: an experimental study using CFRP plates", *Proceedings 8th International Conference on Structural Faults and Repair*, edited by Forde, 12-15 July.

Kachlakev, D. I. (2002), "Finite element analysis and model validation of shear deficient reinforced concrete beams strengthened with GFRP laminate", *3rd Conference on Composites in Infrastructure*, June.

Kachlakev, D. I. (2002), "Finite element analysis of historic bridge strengthened with FRP laminates", *3rd Conference on Composites in Infrastructure*, June.

Kim, S. (2003), "Ductility of carbon fiber reinforced polymer (CFRP) strengthened reinforced concrete beams", Ph.D. Dissertation, Syracuse University, Syracuse, NY.

MacGreger, J. G. (1997), *Reinforced Concrete-Mechanics and Design,* 4rd edition, Prentice Hall, 2005.

Park, S. and Aboutaha, R. S. "*Analysis of CFRP strengthened concrete bridge piers using the STM approach*", Submitted for publication in the *ACI Struct. J.*

CC