New Finite Element for modeling connections in 2D small frames - Static Analysis

Chadi Azoury[1], Assad Kallassy[2], Ibrahim Moukarzel[3]

[1]Mechanical Engineering Department, Lebanese University, Lebanon
[2]Civil Engineering Department, Lebanese University, Lebanon
[3]Faculty of Engineering, Lebanese University, Lebanon

Abstract *— The paper presents a proposal to create a new one-dimensional (1-D) finite element that can substitute for the three-dimensional (3-D) finite model of connection elements in static analysis. The study is conducted on a simple small frame. First, a 3-D model is constructed, followed by the construction of an equivalent two-dimensional (2-D) one. At the final stage, the results of the static analysis of many proposed 1-D element models are compared with the results of the 2-D model. With a special treatment of the connection elements, the results of the analysis of the new 1-D finite element models agree well with the results of the 2-D model.*

Keywords: Structures, Finite element methods, Condensation, Timoshenko beam, Connection element, and Small frames.

Notations

E	Young's elastic modulus
v	Poisson coefficient
ρ	density
I	moment of inertia
E I	flexural beam stiffness = E × I
t	frame thickness
K(*n*×*n*)	stiffness matrix of order *n*
L(*n*×*m*)	rectangular matrix with *n* rows and *m* columns
EB	Euler-Bernoulli
sh	shear deformation
FE	finite element
DOF	degree-of-freedom

I. INTRODUCTION

To perform a static analysis of a frame, 3-D finite elements are usually used in the meshing process. A large number of elements are required, which in turn requires a large number of DOF and consequently, a great deal of work. The main objective consists of developing a 1-D model made up of simple 1-D elements to replace the 3-D model made up of tetrahedral prisms or brick elements. To perform a static analysis, the calculation of the stiffness matrix is required.

In order to find the best 1-D finite meshing element, 3-D meshing is used, and 2-D and 1-D models are consecutively constructed.

All calculations are done by using ANSYS Version 16.2 and MATLAB Version 7. A simple frame is used in the calculations.

II. TEST STRUCTURE

Our test structure will be of this form:

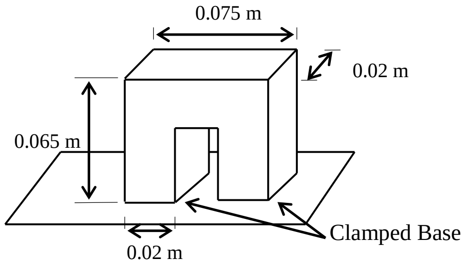


FIGURE 1: TEST STRUCTURE

The beam section of the test structure is rectangular (0.02×0.02 m^2). The mechanical characteristics are: Poisson coefficient $\nu = 0.31$, density $\rho = 78000$ N/m^3, and elastic modulus E = 184 GPa. The test structure is clamped at its base and consists of 5 structural elements shown in Figure 2.

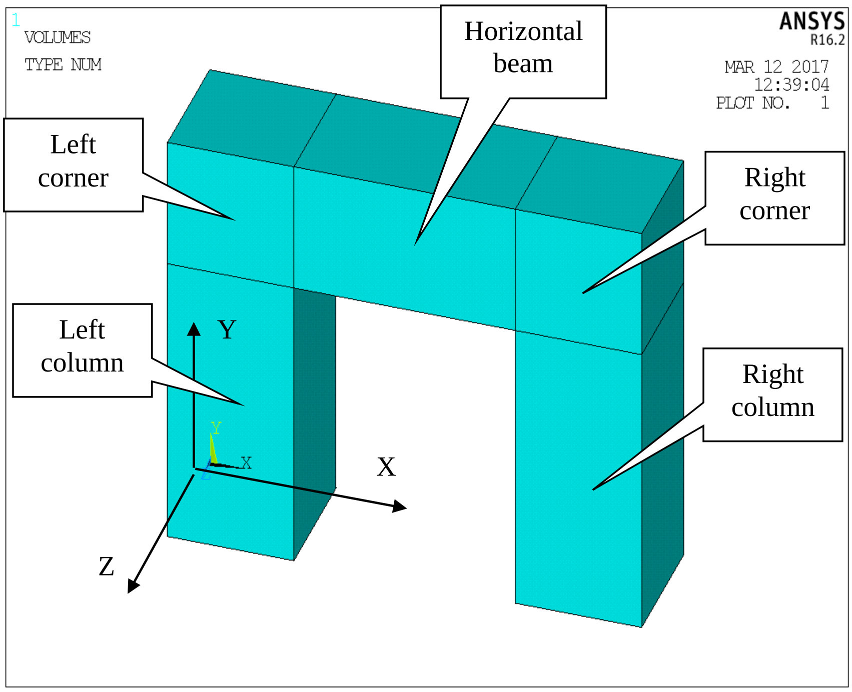


FIGURE 2: TEST STRUCTURE CONSTRUCTED IN ANSYS® R16.2

Different types of elements are used in the meshing processes. In static analysis, the comparison between different models is based on the deformation in the XY plan of the structure. These models are compared in 2 load cases illustrated in

Figure 3:

- Case 1: a body load of 78000 N/m^3 along the vertical axis of the horizontal beam
- Case 2: a body load of 78000 N/m^3 along the horizontal axis of the left column

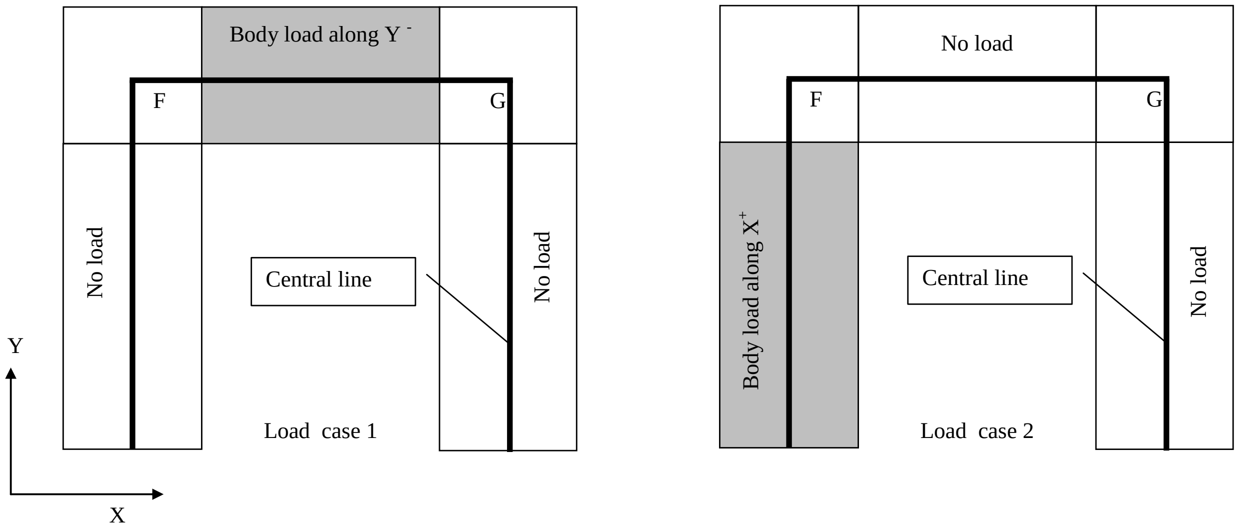


FIGURE 3: TEST STRUCTURE WITH LOAD CASES 1 AND 2

III. 3D AND 2D MODELING

The 3D model of the test structure is meshed using H8 volumetric finite elements with 8 nodes. Each column is modeled using 8×8×16 = 1024 elements and each corner includes 8×8×8 = 512 elements. The horizontal beam is modeled using 8×8×16 = 1024 elements. In total, it is equal to 4096 elements with 5265 nodes (see Figure 4).

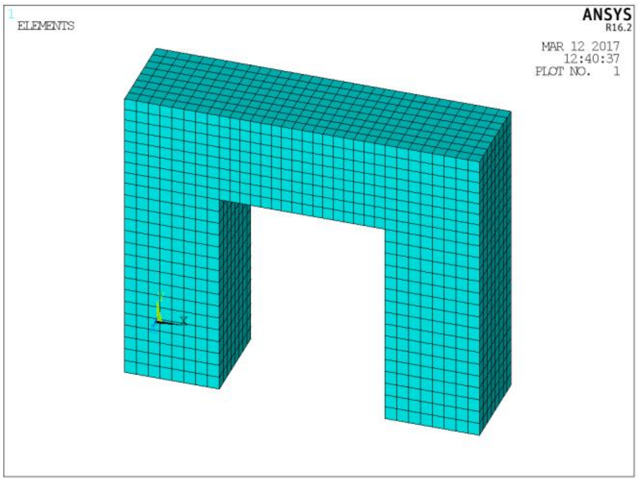


FIGURE 4: 3D TEST STRUCTURE MODELLED USING H8 FINITE ELEMENTS

Figures 5 and 6 show the total displacement contours of the frame under load case 1 and 2.

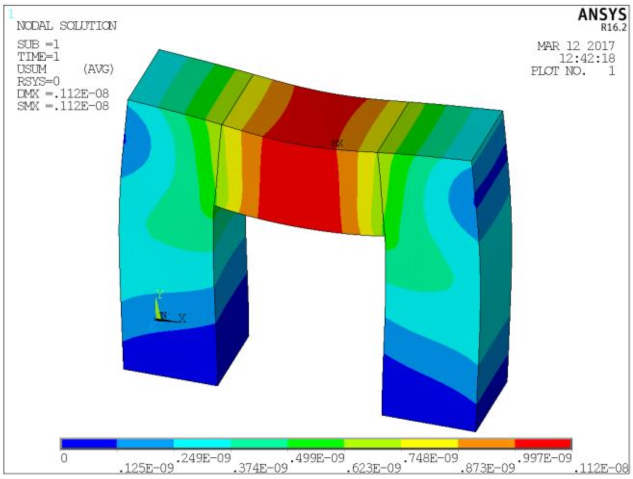


FIGURE 5: THREE-DIMENSIONAL DEFORMED TEST STRUCTURE UNDER LOAD CASE 1

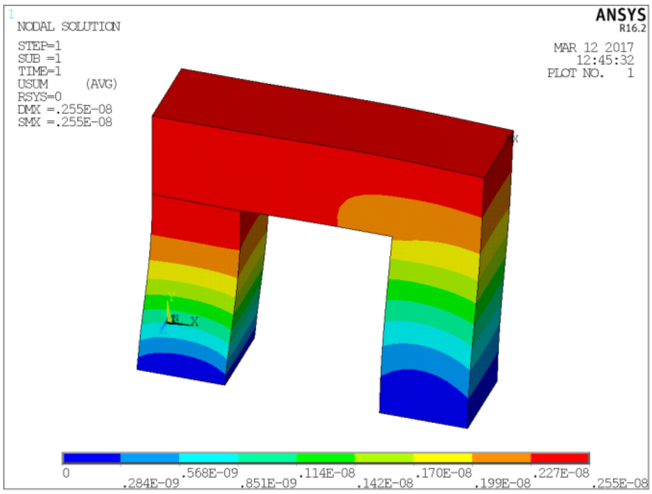


FIGURE 6: THREE-DIMENSIONAL DEFORMED TEST STRUCTURE UNDER LOAD CASE 2

The test structure is then modeled in two dimensions. A Q4 plane element with a thickness t = 0.02 m and having 4 nodes is used. Each column is modeled using 8×16 = 128 elements. Each corner includes 8×8 = 64 elements. The horizontal beam is modeled using 8×16 = 128 elements. The total is 512 elements with 585 nodes (see Figure 9a).

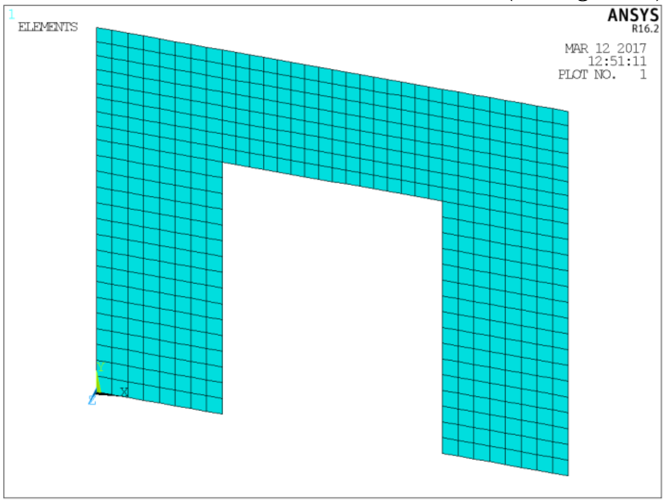


FIGURE 7: 2D TEST STRUCTURE MODELED USING Q4 PLANE FINITE ELEMENTS

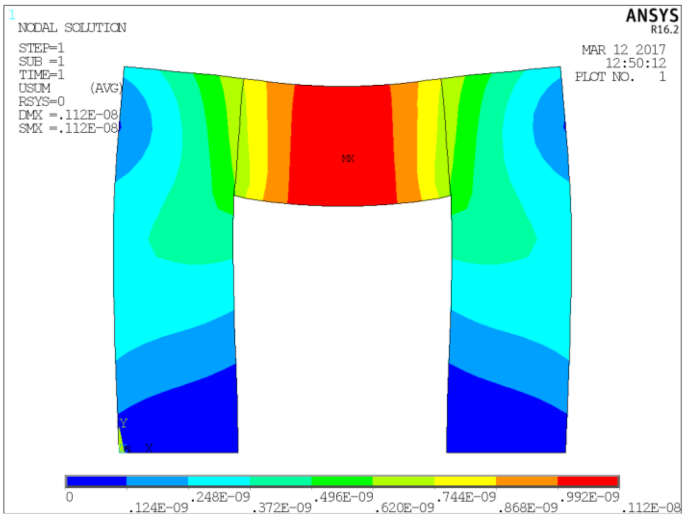


FIGURE 8: TWO-DIMENSIONAL DEFORMED TEST STRUCTURE ALONG LOAD CASE 1

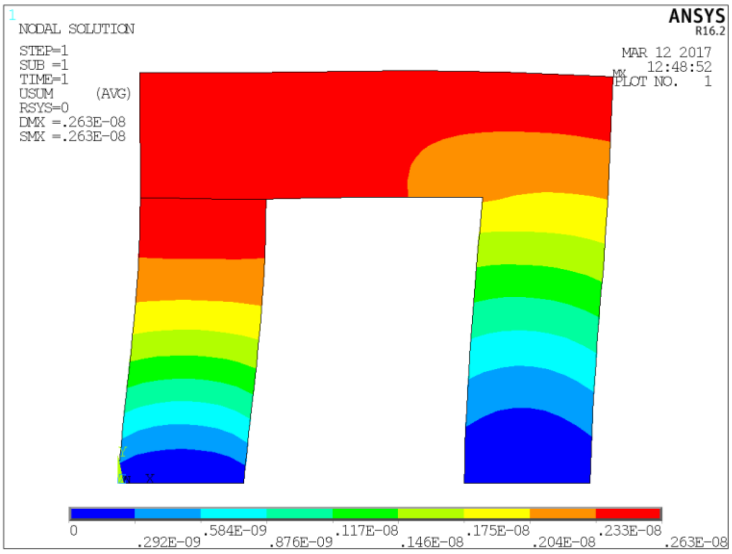


FIGURE 9: TWO-DIMENSIONAL DEFORMED TEST STRUCTURE ALONG LOAD CASE 2

Figures 8 and 9 have similar maps as Figures 5 and 6. The different models are compared using the displacement of 65 points on the central line of the longitudinal symmetrical plan of the structure.

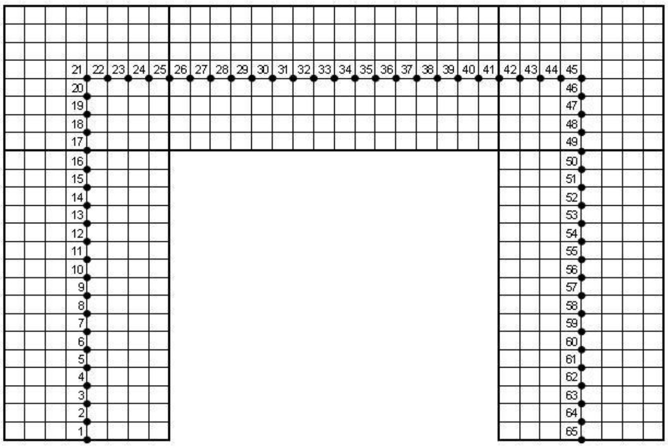


FIGURE 10 : LONGITUDINAL SYMMETRICAL PLAN OF THE STRUCTURE SHOWING THE 65 CENTRAL LINE POINTS

In order to establish a comparison between both FE models and using both load cases, UX and UY displacements of nodes on the central line, along the X and Y axis respectively, are plotted on the same graph (see Figure 11 – 10).

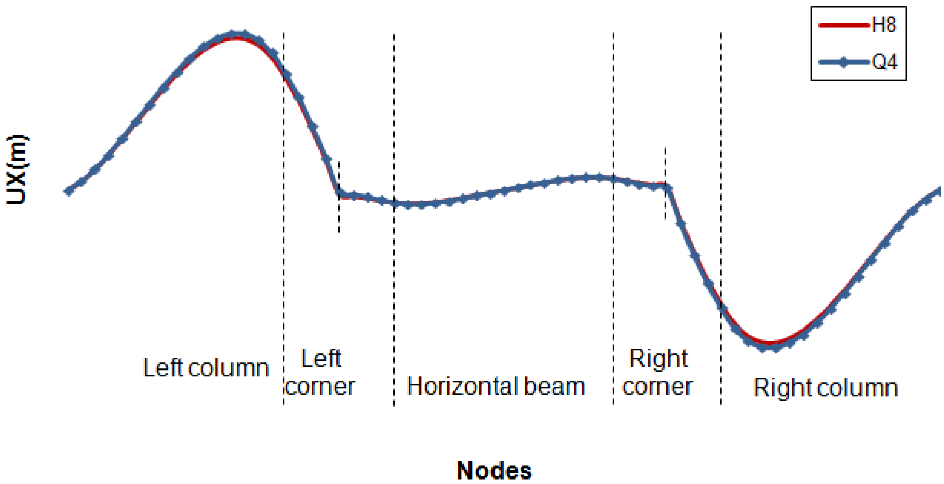


FIGURE 11 : UX (m) CENTRAL LINE DISPLACEMENT ALONG AXE X (LOAD CASE 1)

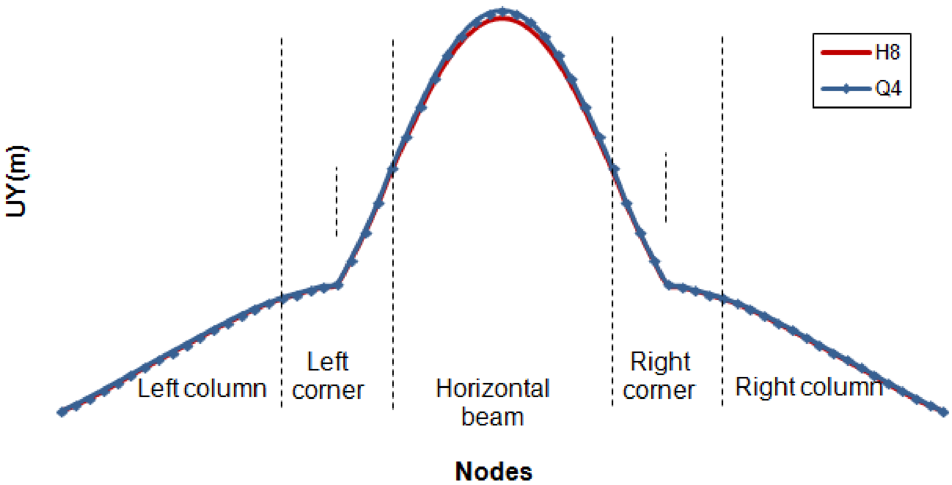


FIGURE 12 : UY (m) CENTRAL LINE DISPLACEMENT ALONG AXE Y (LOAD CASE 1)

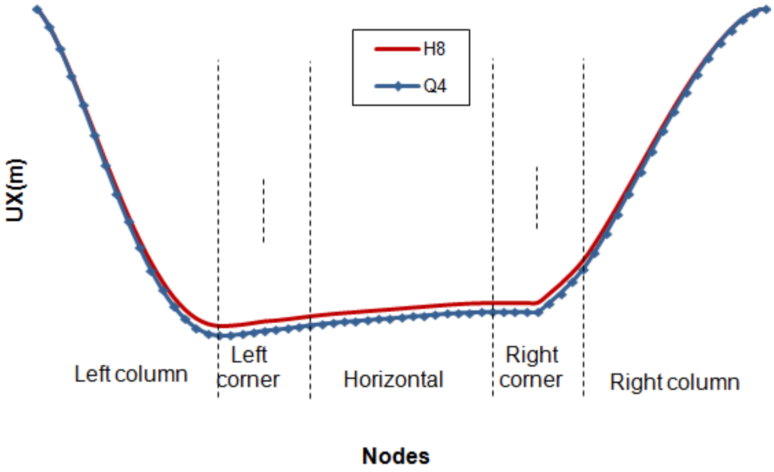


FIGURE 13 : UX (m) CENTRAL LINE DISPLACEMENT ALONG AXE X (LOAD CASE 2)

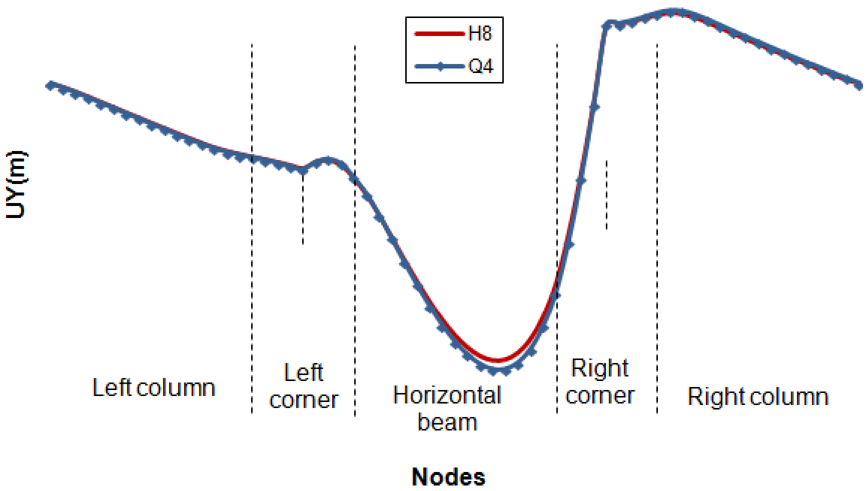


FIGURE 14 : UY (m) CENTRAL LINE DISPLACEMENT ALONG AXE Y (LOAD CASE 2)

Regarding the comparison criteria of the results produced by different models, the following error formula is used:

$$E_{\max} = \max_i \frac{|U_R(i) - U_C(i)|}{\left|\max_j(U_R(j)) - \min_j(U_R(j))\right|}$$

$U_R(i)$ = reference value of a displacement at point i

$U_C(i)$ = calculated value of the same displacement and at the same point i of a new model

The maximum errors for both models H8 and Q4, for both load cases, are listed in Table 1 below:

TABLE 1
COMPARISON BETWEEN H8 AND Q4 MODELS

Load case	Displacement	U_R (m) [H8]	U_C (m) [Q4]	*E*max	Node number (*E*max)
1	UX	-2.6673E-10	-2.7536E-10	1.55%	15 and 51
1	UY	-1.0941E-09	-1.1149E-09	1.90%	33
2	UX	2.4111E-09	2.4842E-09	3.01%	16
2	UY	2.6802E-10	2.7866E-10	2.92%	39

The plane model has produced comparable results to those of the 3-D model. Thus, the reference model is the 2-D model which leads to a reduction in the DOF amount. It will be used to develop the best 1-D element for connection modeling.

IV. ONE-DIMENSIONAL CLASSICAL MODELING

In this section, 1-D elements are used for the test structure. Since the structure is not the slender type, we know that Euler-Bernoulli beams are not adequate. Therefore, we will use Timoshenko beams with 2 nodes per element and 3 DOF per node (Timoshenko model).

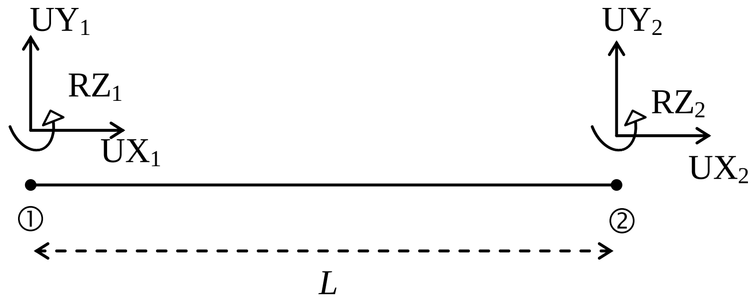


FIGURE 15 : TWO-NODE BEAMS

The transformation from the Q4 model to the 1-D model is shown in Figures 12 and 13. The volumetric charge is transformed into linear charge.

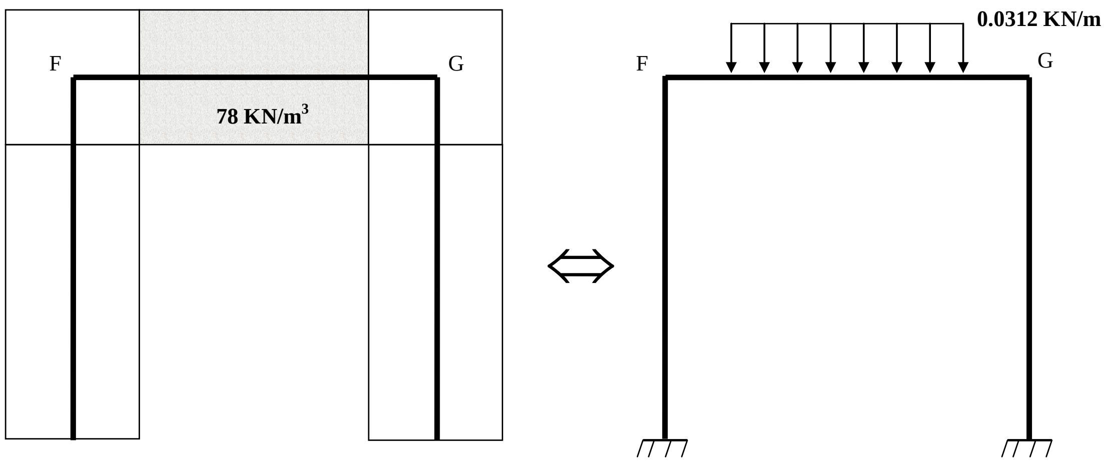


FIGURE 16 : TRANSFORMATION OF THE Q4 MODEL INTO A 1-D MODEL (LOAD CASE 1)

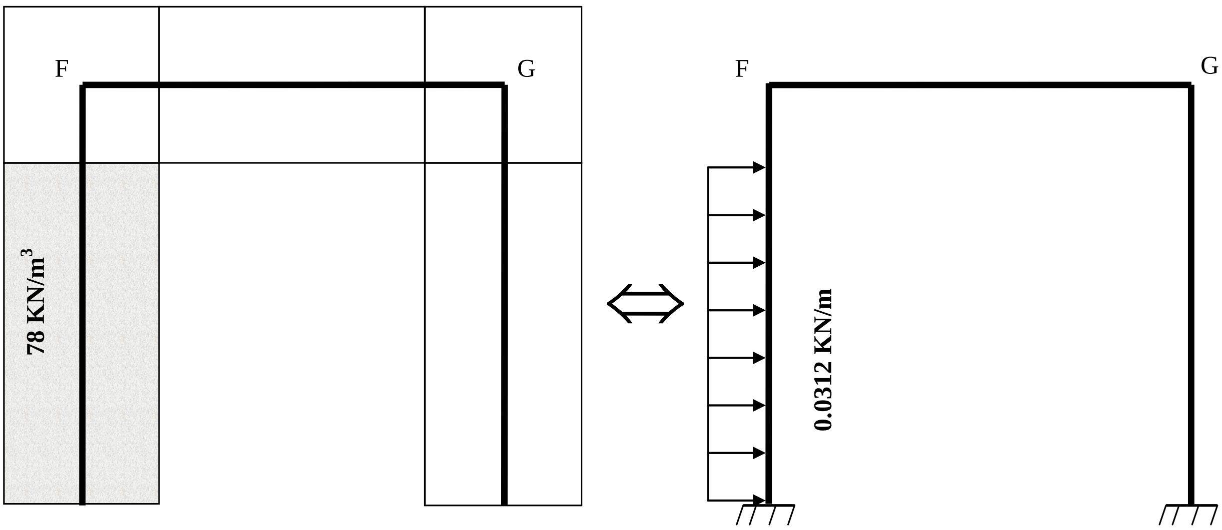


FIGURE 17 : TRANSFORMATION OF THE Q4 MODEL INTO A 1-D MODEL (LOAD CASE 2)

The left and right columns are modeled using 16 elements each. The horizontal beam is also modeled using 16 elements. Both left and right corners are modeled using 4 horizontal linear elements and 4 vertical linear elements, i.e. a total of 64 beam elements. The following figures show the deformed test structure in the 2 load cases as applicable to both models: the reference model Q4 and the Timoshenko model.

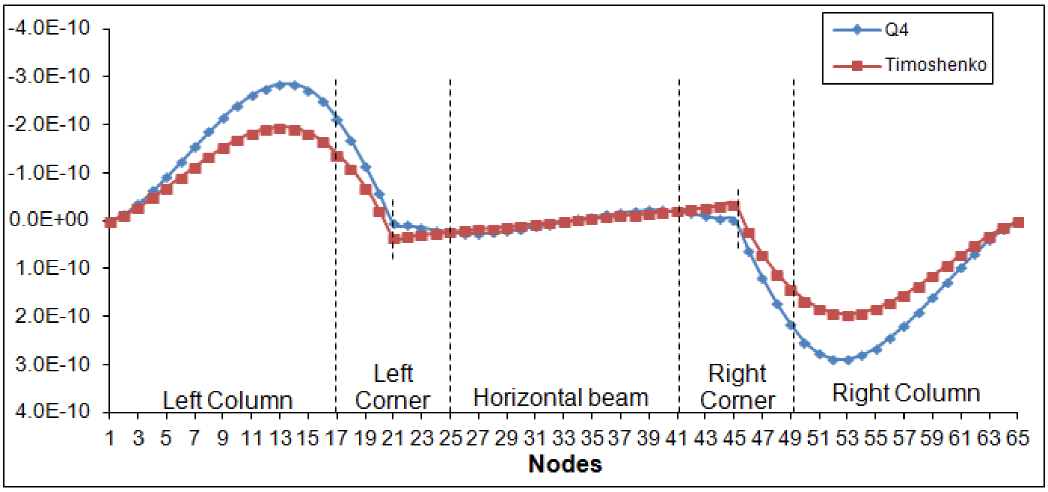


FIGURE 18 : UX (m) DISPLACEMENT OF CENTRAL LINE NODES ALONG X (LOAD CASE 1)

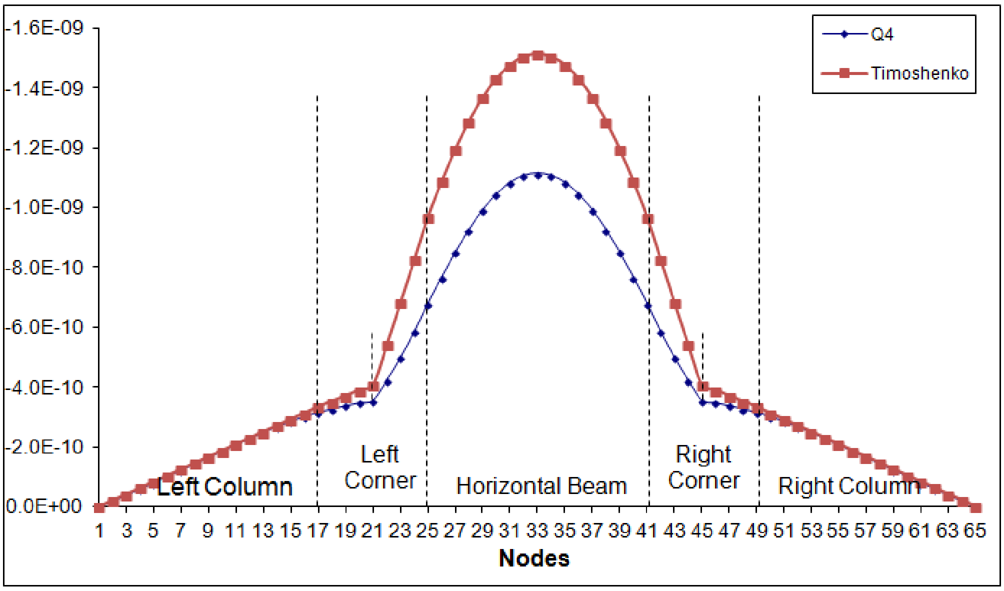


FIGURE 19 : UY (m) DISPLACEMENT OF CENTRAL LINE NODES ALONG Y (LOAD CASE 1)

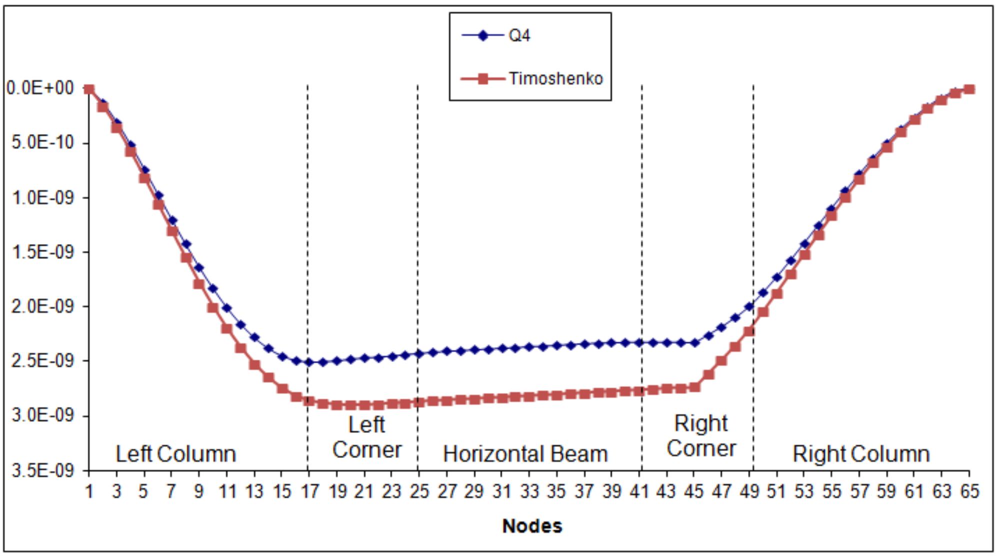


FIGURE 20 : UX (m) DISPLACEMENT OF CENTRAL LINE NODES X (LOAD CASE 2)

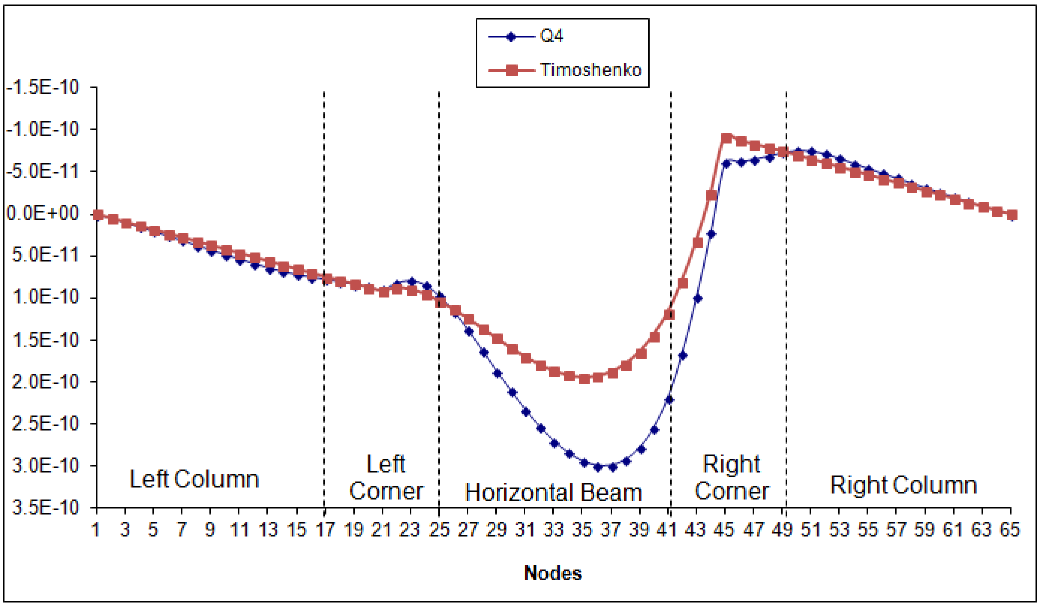


FIGURE 21 : UY (m) DISPLACEMENT OF CENTRAL LINE NODES Y (LOAD CASE 2)

None of the 1-D models produces proximate results in comparison with those given by the Q4 model. Improved models using Timoshenko beams are shown below.

V. FIRST IMPROVED MODEL: CQ CORNER QUAD

A new CQ element is suggested for the corners, based on the utilization of Q4 surface elements since the Q4 modeling and the H8 modeling produce close results. Non-corner elements are still modeled as Timoshenko beams. One way to connect two different meshing types is explained by Dohrmann and al, [4] and [5], and Quiroz and al [9]. Kattner and al [8] explain how modeling joints between frame columns and beams is performed.

The interface between the new CQ element on the one hand and the horizontal and vertical beams on the other hand, is considered rigid in our case. The corner element of 0.02 m × 0.02 m and 0.02 m in thickness is meshed using Q4 elements of 0.02/8 m × 0.02/8 m which produces 64 elements and 81 nodes in total, hence 81×2 DOF: 17 × 2 interface DOF and 64 × 2 non-interface DOF (see next Figure).

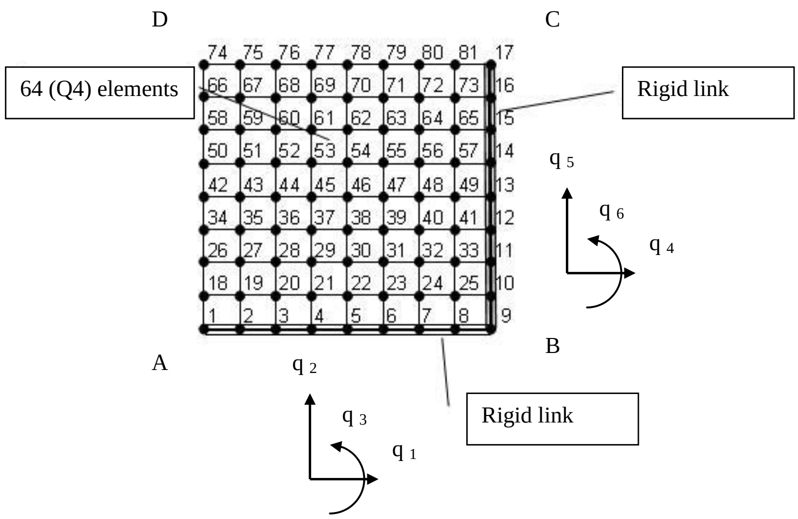


FIGURE 22 : LEFT CORNER WITH RIGID LINKS

The elementary stiffness matrix $k_e(8,8)$ of the Q4 element is mentioned in Cook [2]. By assembling the 64 Q4 elements, the global stiffness matrix of the corner is K(162,162).

Using static condensation, the global stiffness matrix is reduced to $K_c(34,34)$ after elimination of the 128 non-interface DOF [3].

To find matrix Kc(34,34), we use the formula for static condensation. In Matlab code:

```
krr=K(1:34,1:34);
krc=K(1:34,35:162);
kcr=K(35:162,1:34);
kcc=K(35:162,35:162);
Kc=krr-krc*inv(kcc)*kcr;
```

The AB interface is supposed to be rigid. Then, 3 DOF (q_1, q_2, q_3) are enough to determine the DOF of the AB interface:

- q_1: horizontal displacement of the midpoint of segment AB
- q_2: vertical displacement of the midpoint of segment AB
- q_3: rotation of segment AB around its midpoint

The same applies to the BC interface where the DOF (q_4, q_5, q_6) of the midpoint in BC determine the DOF of the rigid interface BC.

The reduced stiffness matrix K_r(6,6) is given by: $K_r = L^T K_c L$, where L(34×6) is the matrix giving the values of the DOF relative to interfaces AB and BC, from degrees q_1, q_2, q_3, q_4, q_5, and q_6.

```
L=[

1   0     0     0     0     0
0   1     -4*a  0     0     0
1   0     0     0     0     0
0   1     -3*a  0     0     0
1   0     0     0     0     0
0   1     -2*a  0     0     0
1   0     0     0     0     0
0   1     -1*a  0     0     0
1   0     0     0     0     0
0   1     0     0     0     0
1   0     0     0     0     0
0   1     1*a   0     0     0
1   0     0     0     0     0
0   1     2*a   0     0     0
1   0     0     0     0     0
0   1     3*a   0     0     0
0   0     0     1     0     4*b
0   1     4*a   0     0     0
0   0     0     1     0     3*b
0   0     0     0     1     0
0   0     0     1     0     2*b
0   0     0     0     1     0
0   0     0     1     0     1*b
0   0     0     0     1     0
0   0     0     1     0     0
0   0     0     0     1     0
0   0     0     1     0     -1*b
0   0     0     0     1     0
0   0     0     1     0     -2*b
0   0     0     0     1     0
0   0     0     1     0     -3*b
0   0     0     0     1     0
0   0     0     1     0     -4*b
0   0     0     0     1     0     ];
```

The stiffness matrix K_r(6×6) is a function of:

- The length a of side AB
- The length b of side BC

- The elastic modulus E of the material in part ABCD
- The Poisson coefficient *v*
- The thickness of the frame t

The stiffness matrix of the left corner beam of the test structure is given numerically by:

```
kleft=[
  4.6660e+009  1.5022e+009 -5.2058e+006 -4.6660e+009 -1.5022e+009 -2.6432e+007
  1.5022e+009  4.6660e+009  2.6432e+007 -1.5022e+009 -4.6660e+009  5.2058e+006
 -5.2058e+006  2.6432e+007  3.7398e+005  5.2058e+006 -2.6432e+007 -5.7599e+004
 -4.6660e+009 -1.5022e+009  5.2058e+006  4.6660e+009  1.5022e+009  2.6432e+007
 -1.5022e+009 -4.6660e+009 -2.6432e+007  1.5022e+009  4.6660e+009 -5.2058e+006
 -2.6432e+007  5.2058e+006 -5.7599e+004  2.6432e+007 -5.2058e+006  3.7398e+005]
```

With a simple transformation, we can obtain the stiffness matrix of the right corner beam:

```
kright=[
  4.6660e+009 -1.5022e+009 -5.2058e+006 -4.6660e+009  1.5022e+009 -2.6432e+007
 -1.5022e+009  4.6660e+009 -2.6432e+007  1.5022e+009 -4.6660e+009 -5.2058e+006
 -5.2058e+006 -2.6432e+007  3.7398e+005  5.2058e+006  2.6432e+007 -5.7599e+004
 -4.6660e+009  1.5022e+009  5.2058e+006  4.6660e+009 -1.5022e+009  2.6432e+007
  1.5022e+009 -4.6660e+009  2.6432e+007 -1.5022e+009  4.6660e+009  5.2058e+006
 -2.6432e+007 -5.2058e+006 -5.7599e+004  2.6432e+007  5.2058e+006  3.7398e+005]
```

Finally, the CQ model of the constructed frame is as follows:

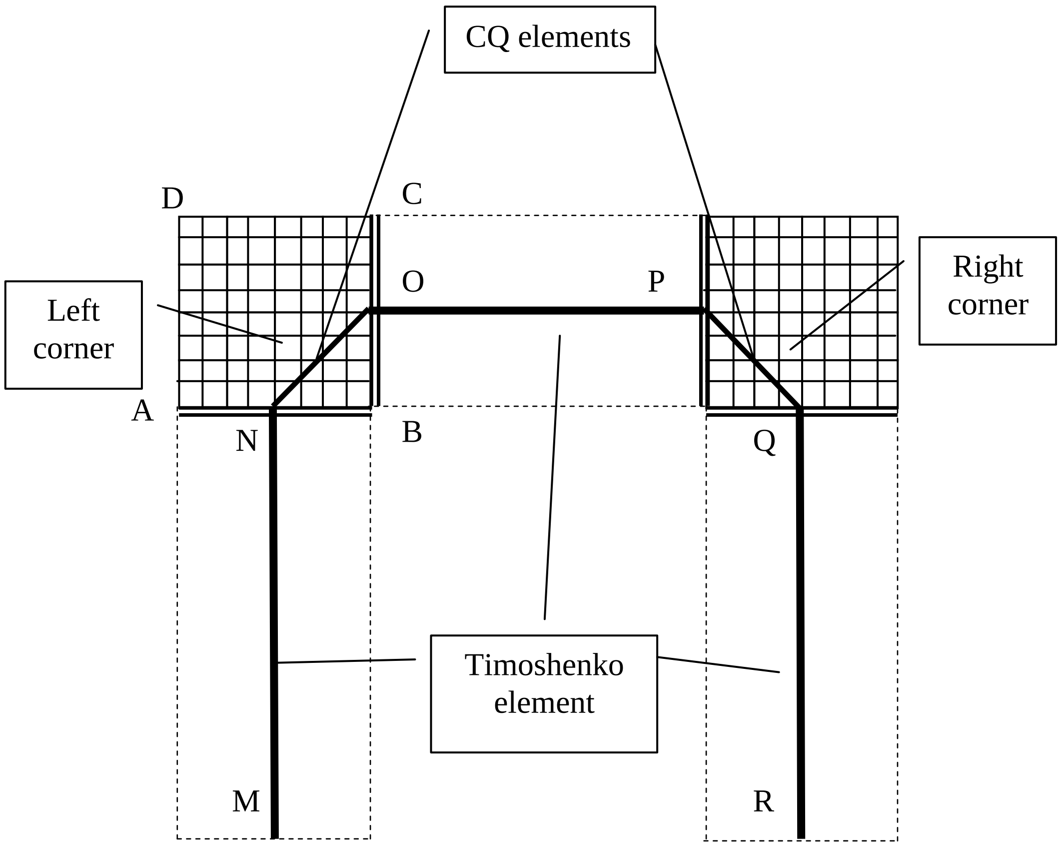


FIGURE 23 : DESCRIPTION OF THE CQ MODEL

- MN, OP and QR are Timoshenko beams
- NO is a CQ element with a $K_{r\ left}(6\times6)$ stiffness matrix
- PQ is a CQ element with a $K_{r\ right}(6\times6)$ stiffness matrix

Figures 24 to 27 show static results under load cases 1 and 2.

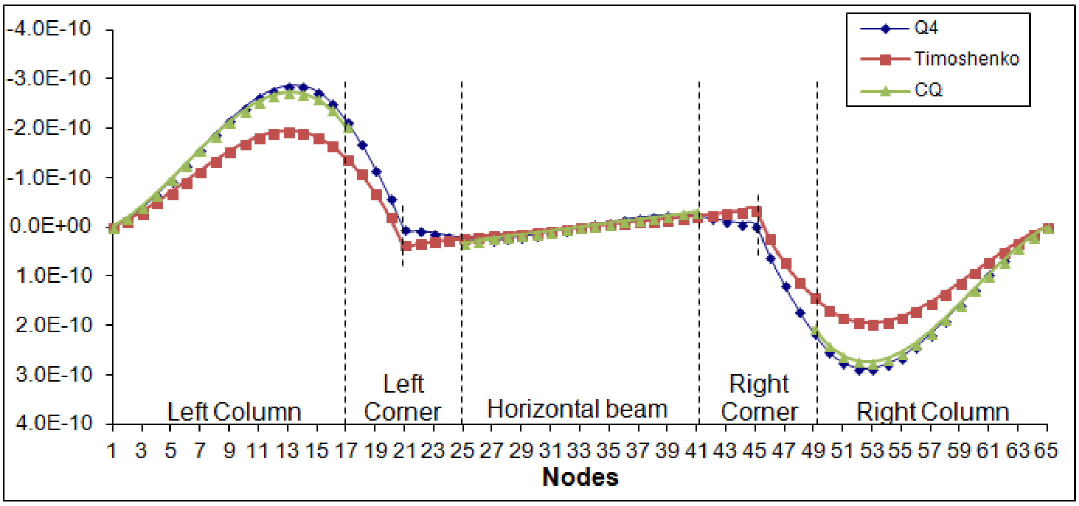


FIGURE 24 : UX (m) DISPLACEMENT OF CENTRAL LINE NODES ALONG X (LOAD CASE 1)

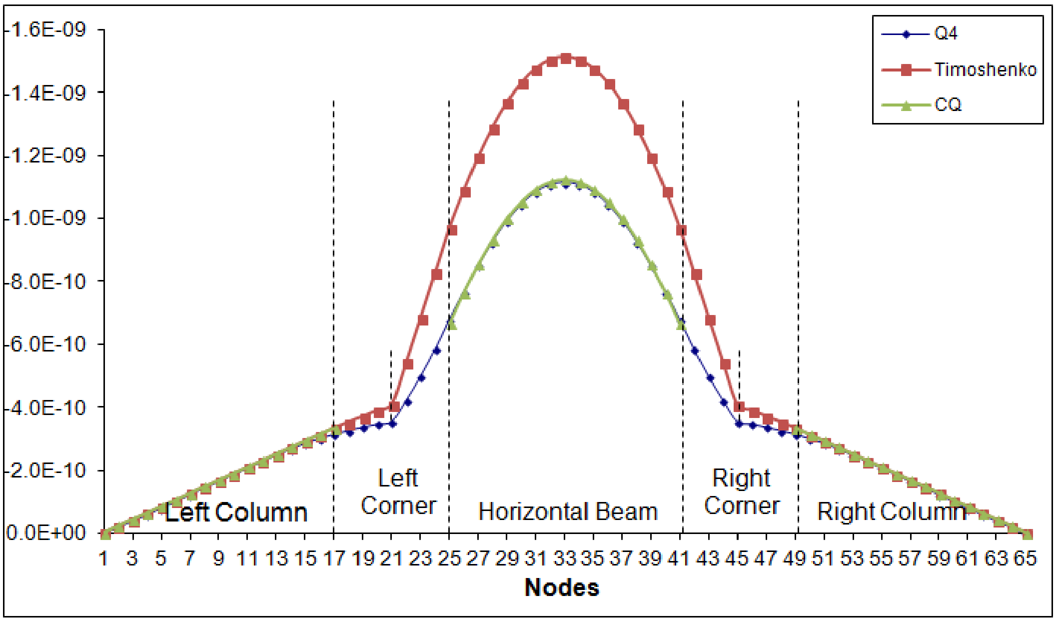


FIGURE 25 : UY (m) DISPLACEMENT OF THE CENTRAL LINE NODES ALONG Y (LOAD CASE 1)

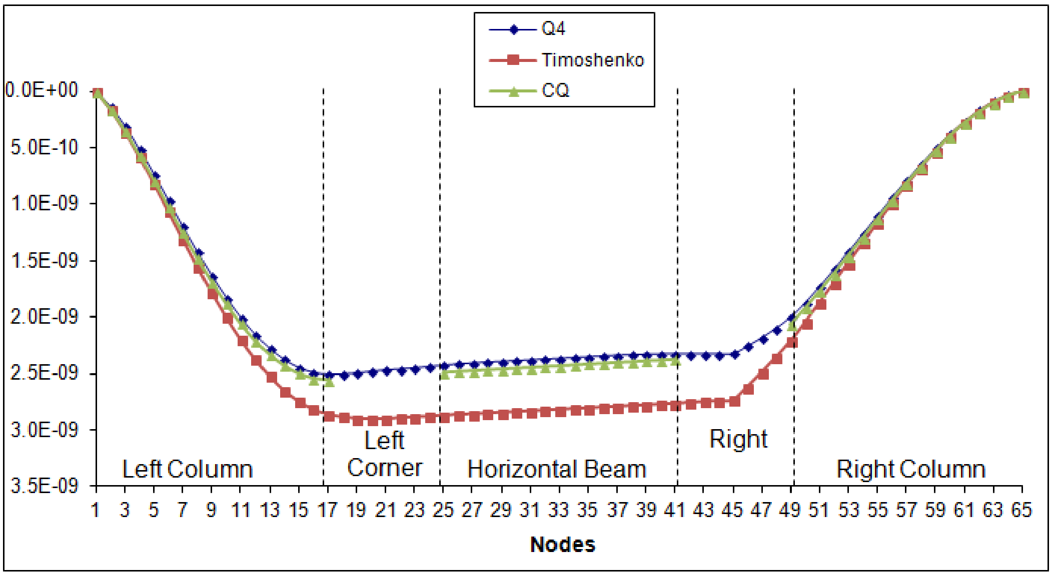


FIGURE 26 : UX (m) DISPLACEMENT OF THE CENTRAL LINE NODES ALONG X (LOAD CASE 2)

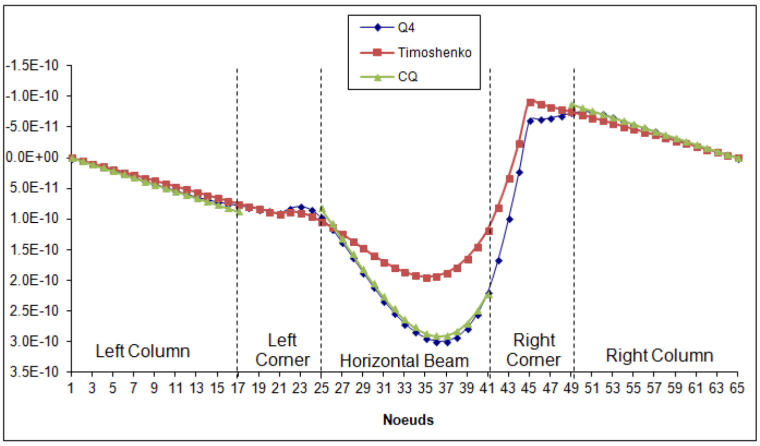


FIGURE 27 : UY (m) DISPLACEMENT OF THE CENTRAL LINE NODES ALONG Y (LOAD CASE 2)

The graphs show that the results of the CQ model are very close to those of the Q4 model. Table 2 shows the results of both models for both load cases. The last column gives the node number corresponding to the highest error value E_{max}.

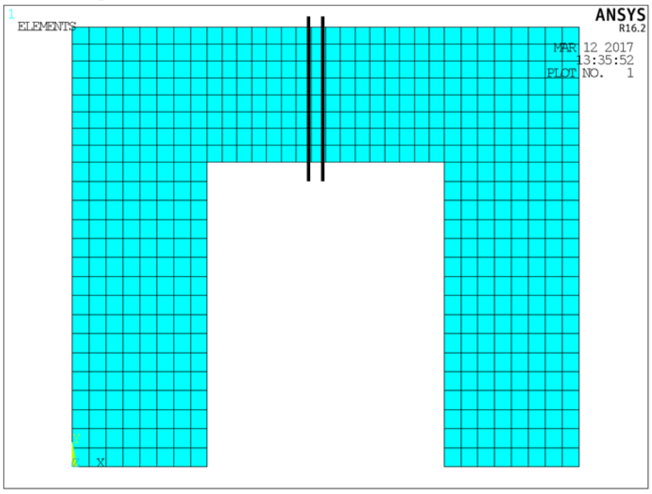
TABLE 2
COMPARISON OF BOTH MODELS AND ALL LOAD CASES

Load case	Displacement	U_R (m) [Q4]	Model	U_C (m)	E_{max}	Node number (E_{max})
1	UX	2.8613E-10	Timoshenko	1.9284E-10	16.28%	14 or 52
1	UY	-1.1149E-09	Timoshenko	-1.5148E-09	35.87%	33
2	UX	2.3397E-09	Timoshenko	2.7944E-09	18.19%	36 or 37
2	UY	2.7866E-10	Timoshenko	1.645E-10	30.38%	39
1	**UX**	-2.7536E-10	**CQ**	-2.6053E-10	**2.59%**	**15 or 51**
1	**UY**	-3.1495E-10	**CQ**	-3.3374E-10	**1.69%**	**17 or 49**
2	**UX**	2.3947E-09	**CQ**	2.4623E-09	**2.70%**	**28**
2	**UY**	9.7806E-11	**CQ**	8.2391E-11	**4.10%**	**25**

It can be said from the previous table, that the CQ model produces the best approximation to the Q4 model.

VI. SECOND IMPROVED MODEL: RIGID INTERFACE EVERYWHERE (RQ)

In this paragraph, we propose a linear finite element to replace Timoshenko beams in the vertical and horizontal beams. This new element is constructed using the same procedure for the corner elements as in the previous paragraph.
Consider a strip in the beam that is between two consecutive nodes of the midline (see next figure).



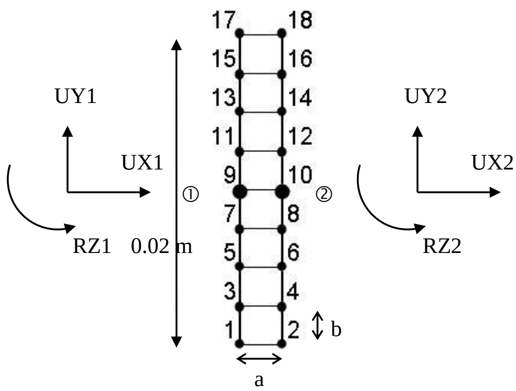


FIGURE 28: RQ ELEMENT

The total length of this element is 0.02 m (from bottom to top). The thickness (along z direction) is always 0.02 m. a = b = 0.045/16 = 0.0028125 m.

The element is discretized in 8 Q4 elements along its length. Therefore, we have a total of 9 nodes and 18 DOF for each interface.

We will assume that plane sections of beams in bending (Timoshenko hypothesis), displacements of nodes that are in the same cut-section are displacements of a rigid body, given by 3 DOFs:

- UX = Displacement along the X axis of the center of the section
- UY = Displacement along the Y axis of the center of the section
- RZ = Rotation around the Z axis

Therefore, the displacements of all the nodes of this element are given by 6 DOFs; 3 DOFs for the left cut-section and 3 DOFs for the right cut-section. This will give us the elementary stiffness matrix ke(6,6). In order to determine ke(6,6), we know that the stiffness matrix for one Q4 element is given by:

The global matrix is K(36,36).

The matrix of the rigid interface is given by:

```
R=[
1  0     4*b   0     0     0
0  1     0     0     0     0
0  0     0     1     0     4*b
0  0     0     0     1     0
1  0     3*b   0     0     0
0  1     0     0     0     0
0  0     0     1     0     3*b
0  0     0     0     1     0
1  0     2*b   0     0     0
0  1     0     0     0     0
0  0     0     1     0     2*b
0  0     0     0     1     0
1  0     1*b   0     0     0
0  1     0     0     0     0
0  0     0     1     0     1*b
0  0     0     0     1     0
1  0     0     0     0     0
0  1     0     0     0     0
0  0     0     1     0     0
0  0     0     0     1     0
1  0     -1*b  0     0     0
0  1     0     0     0     0
0  0     0     1     0     -1*b
0  0     0     0     1     0
1  0     -2*b  0     0     0
0  1     0     0     0     0
0  0     0     1     0     -2*b
0  0     0     0     1     0
1  0     -3*b  0     0     0
0  1     0     0     0     0
0  0     0     1     0     -3*b
0  0     0     0     1     0
1  0     -4*b  0     0     0
0  1     0     0     0     0
0  0     0     1     0     -4*b
0  0     0     0     1     0     ]
```

Therefore, $K = L^T \times Kc \times L$. (Kc is the condensed matrix). We obtain:

```
k=E*t/(v^2-1)*b/a*[
-8,        0,                           0, 8,        0,                    0
 0,    4*v-4,                   2*a*(v-1), 0,    -4*v+4,           2*a*(v-1)
 0,2*a*(v-1),-128/3*b^2-4/3*a^2+4/3*a^2*v, 0,-2*a*(v-1), 128/3*b^2-2/3*a^2+2/3*a^2*v
 8,        0,                           0,-8,        0,                    0
 0,   -4*v+4,                  -2*a*(v-1), 0,     4*v-4,          -2*a*(v-1)
 0,2*a*(v-1), 128/3*b^2-2/3*a^2+2/3*a^2*v, 0,-2*a*(v-1),-128/3*b^2-4/3*a^2+4/3*a^2*v]
```

Our new modeling is shown in the next figure:

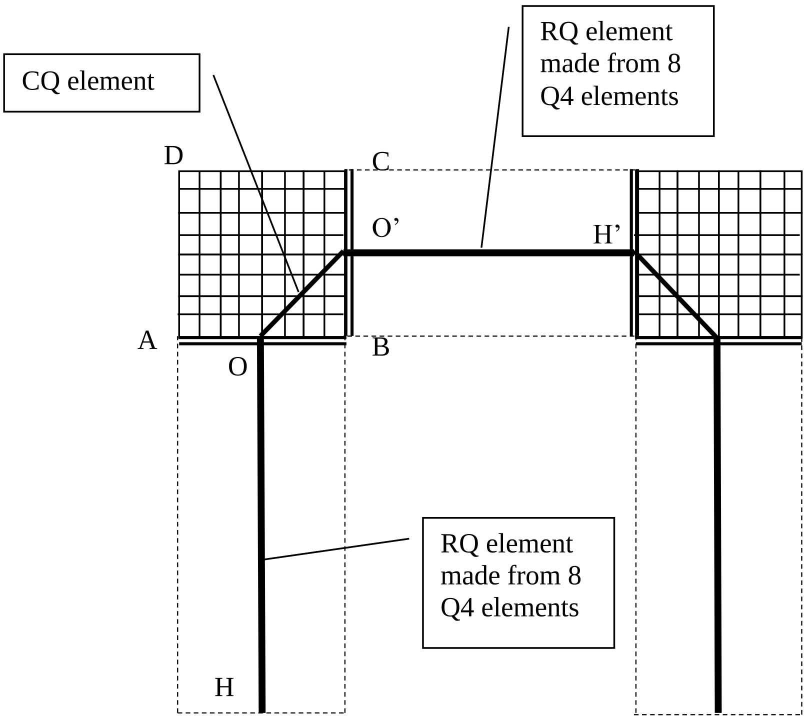


FIGURE 29: RQ MODEL FOR THE HORIZONTAL AND VERTICAL BEAMS WHILE CQ ELEMENT AT THE CORNERS

HO and O'H' are beam elements formed from 8 Q4 elements. OO' is a corner element.

The following graphs show the displacements along X and Y, for the two load cases for all the models that we have considered so far (Q4, Timoshenko, CQ, and RQ):

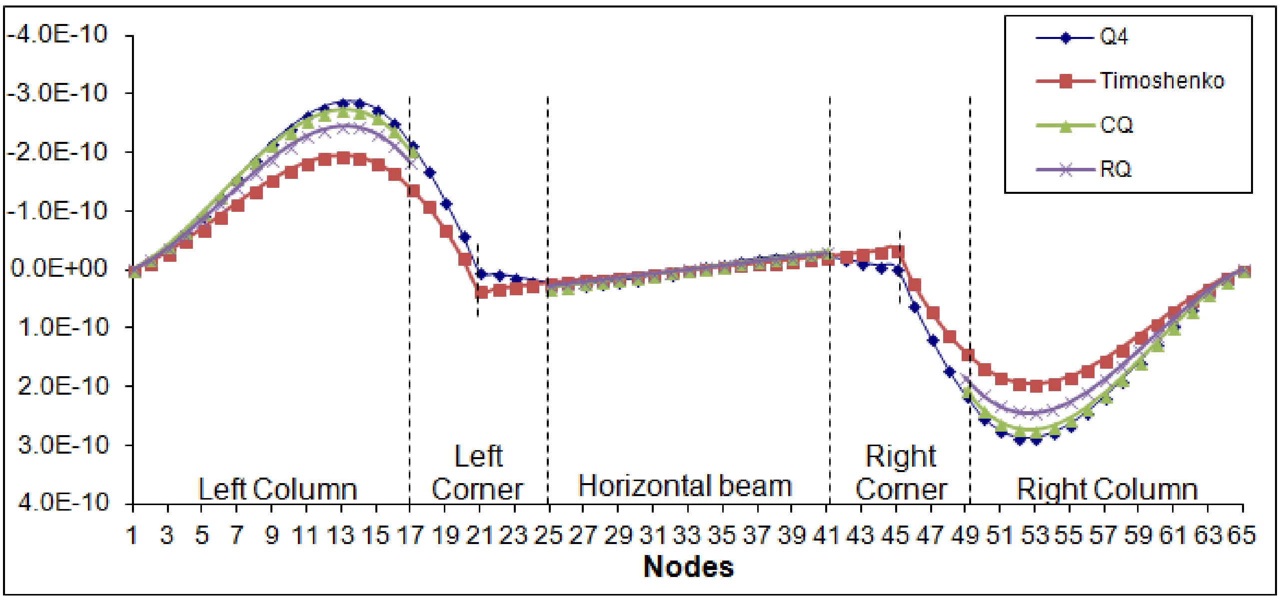


FIGURE 30 : UX (m) DISPLACEMENT OF THE CENTRAL LINE NODES ALONG X (LOAD CASE 1)

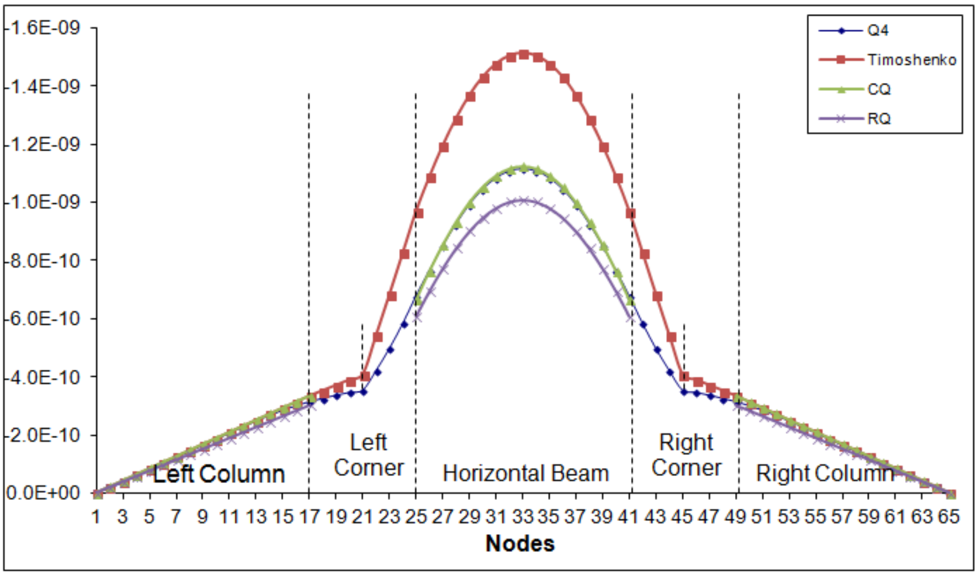


FIGURE 31 : UY (m) DISPLACEMENT OF THE CENTRAL LINE NODES ALONG Y (LOAD CASE 1)

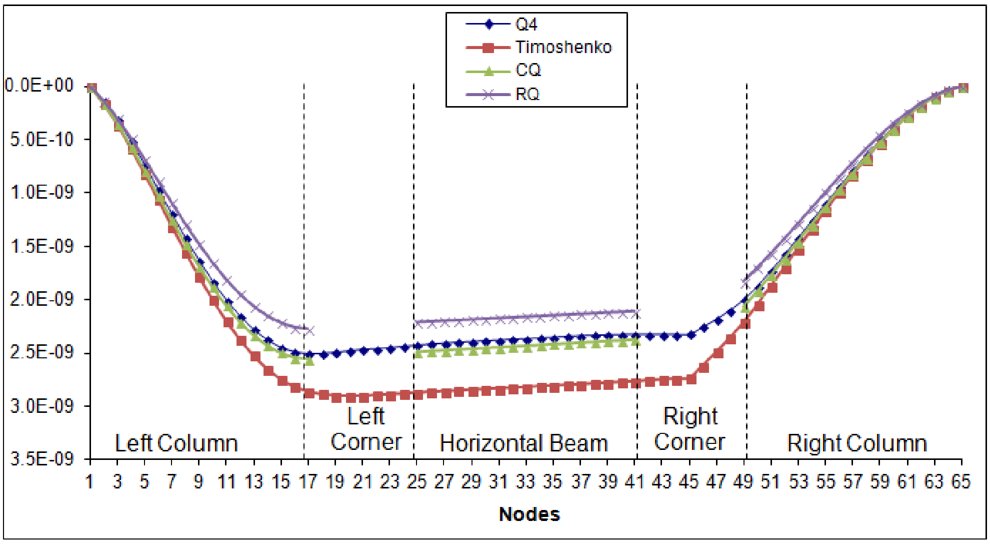


FIGURE 32: UX (m) DISPLACEMENT OF THE CENTRAL LINE NODES ALONG X (LOAD CASE 2)

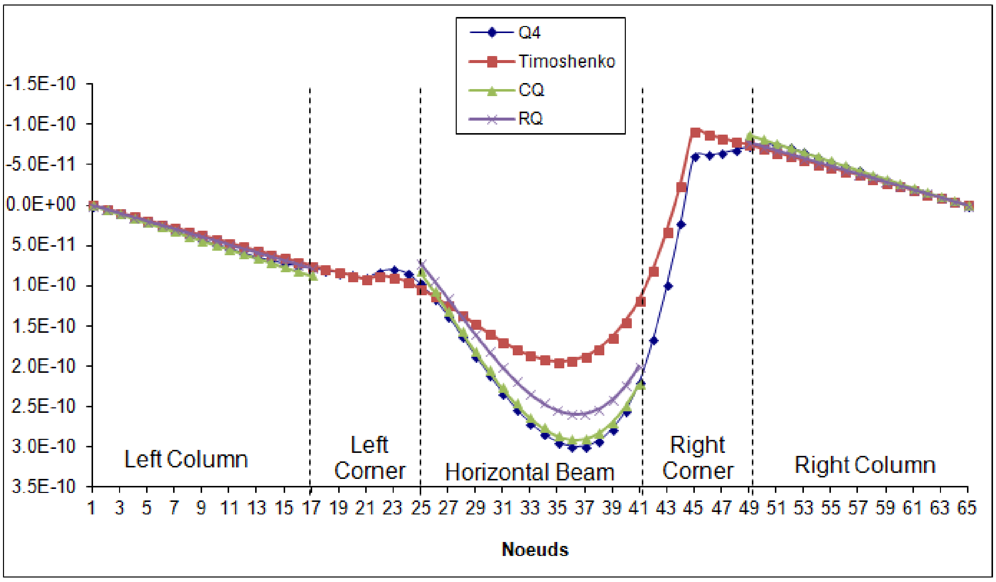


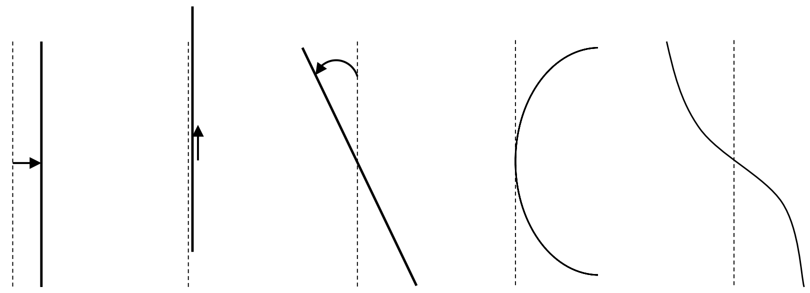
FIGURE 33 : UY (m) DISPLACEMENT OF THE CENTRAL LINE NODES ALONG Y (LOAD CASE 2)

VII. FOURTH IMPROVED MODEL (RQw)

The graphs of the previous paragraph show that the RQ model is a little stiffer than the CQ model. In order to give it a certain flexibility, we will add two DOFs (warping) for every node of the previous paragraph:

A new element is constructed from the RQ model that allows a plane section of a beam to deform a cubic or quadratic deformation after solicitation. Therefore, two DOFs will be added on each node:

A warping (in terms of y^2) that allows a quadratic displacement of the plane section and another warping term (in terms of y^3) that allows a cubic displacement of the plane section. The following figure illustrates the 5 DOFs on each node for the new element.



1st DOF Translation UX | 2nd DOF Translation UY | 3rd DOF Rotation RZ | 4th DOF Warp y^2 | 5th DOF Warp y^3

FIGURE 34: THE 5 DOFS FOR THE RQW MODEL

For this element with 5 DOFs per node we expect a stiffness matrix k(10,10). The terms of the 4th and 5th degree of the stiffness degree are calculated as follows:

We find the global matrix K(36,36), then matrix L of the rigid link:

```
L=[
1       0       4*b  -16*b^2          64*b^3  0       0       0       0       0
0       1       0          0          0       0       0       0       0       0
0       0       0          0          0       1       0       4*b     -16*b^2 64*b^3
0       0       0          0          0       0       1       0       0       0
1       0       3*b  -9*b^2           27*b^3  0       0       0       0       0
0       1       0          0          0       0       0       0       0       0
0       0       0          0          0       1       0       3*b     -9*b^2  27*b^3
0       0       0          0          0       0       1       0       0       0
1       0       2*b  -4*b^2           8*b^3   0       0       0       0       0
0       1       0          0          0       0       0       0       0       0
0       0       0          0          0       1       0       2*b     -4*b^2  8*b^3
0       0       0          0          0       0       1       0       0       0
1       0       1*b  -1*b^2           1*b^3   0       0       0       0       0
0       1       0          0          0       0       0       0       0       0
0       0       0          0          0       1       0       1*b     -1*b^2  1*b^3
0       0       0          0          0       0       1       0       0       0
1       0       0          0          0       0       0       0       0       0
0       1       0          0          0       0       0       0       0       0
0       0       0          0          0       1       0       0       0       0
0       0       0          0          0       0       1       0       0       0
1       0       -1*b    -1*b^2        -1*b^3  0       0       0       0       0
0       1       0          0          0       0       0       0       0       0
0       0       0          0          0       1       0       -1*b    -1*b^2  -1*b^3
0       0       0          0          0       0       1       0       0       0
1       0       -2*b -4*b^2           -8*b^3  0       0       0       0       0
0       1       0          0          0       0       0       0       0       0
0       0       0          0          0       1       0       -2*b    -4*b^2  -8*b^3
0       0       0          0          0       0       1       0       0       0
1       0       -3*b    -9*b^2        -27*b^3 0       0       0       0       0
0       1       0          0          0       0       0       0       0       0
0       0       0          0          0       1       0       -3*b    -9*b^2  -27*b^3
0       0       0          0          0       0       1       0       0       0
1       0       -4*b    -16*b^2       -64*b^3 0       0       0       0       0
0       1       0          0          0       0       0       0       0       0
0       0       0          0          0       1       0       -4*b    -16*b^2 -64*b^3
0       0       0          0          0       0       1       0       0       0       ];
```

We can find:

```
k = L^T*K*L
```

the stiffness matrix k for the left corner with (warp) is given numerically by:

```
kleft =
  4.6660e+009  1.5022e+009 -5.2058e+006 -7.3236e+004 -4.3907e+002 -4.6660e+009 -1.5022e+009 -2.6432e+007  2.5186e+005 -1.8964e+003
  1.5022e+009  4.6660e+009  2.6432e+007 -2.5186e+005  1.8964e+003 -1.5022e+009 -4.6660e+009  5.2058e+006  7.3236e+004  4.3907e+002
 -5.2058e+006  2.6432e+007  3.7398e+005 -1.6755e+003  2.7583e+001  5.2058e+006 -2.6432e+007 -5.7599e+004 -1.1072e+002 -4.2280e+000
 -7.3236e+004 -2.5186e+005 -1.6755e+003  2.8150e+001 -1.2750e-001  7.3236e+004  2.5186e+005 -1.1072e+002 -4.7061e+000  8.7138e-005
 -4.3907e+002  1.8964e+003  2.7583e+001 -1.2750e-001  2.6159e-003  4.3907e+002 -1.8964e+003 -4.2280e+000  8.7138e-005 -4.0191e-004
 -4.6660e+009 -1.5022e+009  5.2058e+006  7.3236e+004  4.3907e+002  4.6660e+009  1.5022e+009  2.6432e+007 -2.5186e+005  1.8964e+003
 -1.5022e+009 -4.6660e+009 -2.6432e+007  2.5186e+005 -1.8964e+003  1.5022e+009  4.6660e+009 -5.2058e+006 -7.3236e+004 -4.3907e+002
 -2.6432e+007  5.2058e+006 -5.7599e+004 -1.1072e+002 -4.2280e+000  2.6432e+007 -5.2058e+006  3.7398e+005 -1.6755e+003  2.7583e+001
  2.5186e+005  7.3236e+004 -1.1072e+002 -4.7061e+000  8.7138e-005 -2.5186e+005 -7.3236e+004 -1.6755e+003  2.8150e+001 -1.2750e-001
 -1.8964e+003  4.3907e+002 -4.2280e+000  8.7138e-005 -4.0191e-004  1.8964e+003 -4.3907e+002  2.7583e+001 -1.2750e-001  2.6159e-003
```

With a simple transformation, we can find the stiffness matrix of the right corner of our structure:

```
kright =
  4.6660e+009 -1.5022e+009 -5.2058e+006 -7.3236e+004 -4.3907e+002 -4.6660e+009  1.5022e+009 -2.6432e+007  2.5186e+005 -1.8964e+003
 -1.5022e+009  4.6660e+009 -2.6432e+007  2.5186e+005 -1.8964e+003  1.5022e+009 -4.6660e+009 -5.2058e+006 -7.3236e+004 -4.3907e+002
 -5.2058e+006 -2.6432e+007  3.7398e+005 -1.6755e+003  2.7583e+001  5.2058e+006  2.6432e+007 -5.7599e+004 -1.1072e+002 -4.2280e+000
 -7.3236e+004  2.5186e+005 -1.6755e+003  2.8150e+001 -1.2750e-001  7.3236e+004 -2.5186e+005 -1.1072e+002 -4.7061e+000  8.7138e-005
 -4.3907e+002 -1.8964e+003  2.7583e+001 -1.2750e-001  2.6159e-003  4.3907e+002  1.8964e+003 -4.2280e+000  8.7138e-005 -4.0191e-004
 -4.6660e+009  1.5022e+009  5.2058e+006  7.3236e+004  4.3907e+002  4.6660e+009 -1.5022e+009  2.6432e+007 -2.5186e+005  1.8964e+003
  1.5022e+009 -4.6660e+009  2.6432e+007 -2.5186e+005  1.8964e+003 -1.5022e+009  4.6660e+009  5.2058e+006  7.3236e+004  4.3907e+002
 -2.6432e+007 -5.2058e+006 -5.7599e+004 -1.1072e+002 -4.2280e+000  2.6432e+007  5.2058e+006  3.7398e+005 -1.6755e+003  2.7583e+001
  2.5186e+005 -7.3236e+004 -1.1072e+002 -4.7061e+000  8.7138e-005 -2.5186e+005  7.3236e+004 -1.6755e+003  2.8150e+001 -1.2750e-001
 -1.8964e+003 -4.3907e+002 -4.2280e+000  8.7138e-005 -4.0191e-004  1.8964e+003  4.3907e+002  2.7583e+001 -1.2750e-001  2.6159e-003
```

The following figures show the displacements along X et Y, for both the load cases, for all models (Reference model: Q4 model, Timoshenko model, CQ model, RQ, model and RQw).

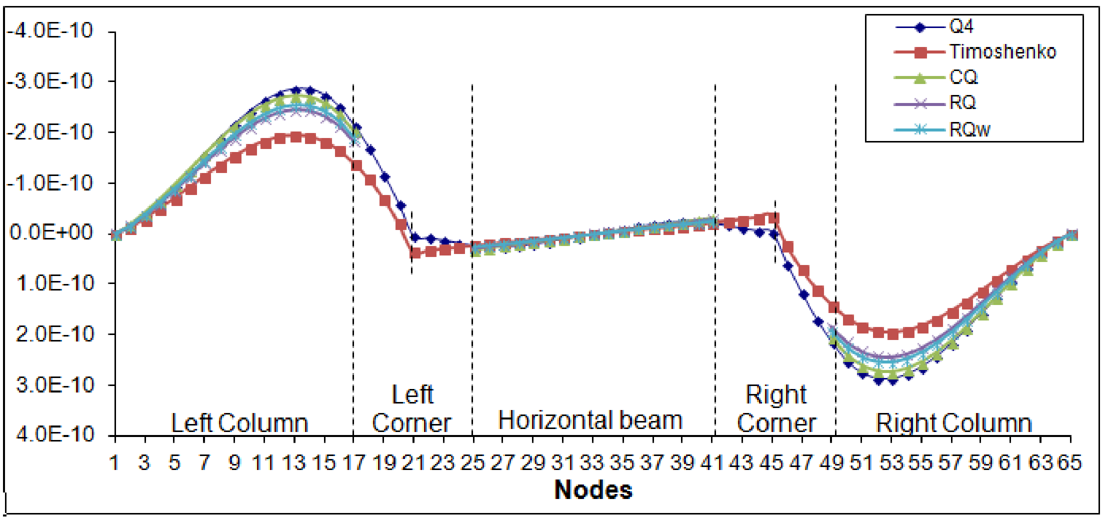


FIGURE 35: UX (m) DISPLACEMENT OF THE CENTRAL LINE NODES ALONG X (LOAD CASE 1)

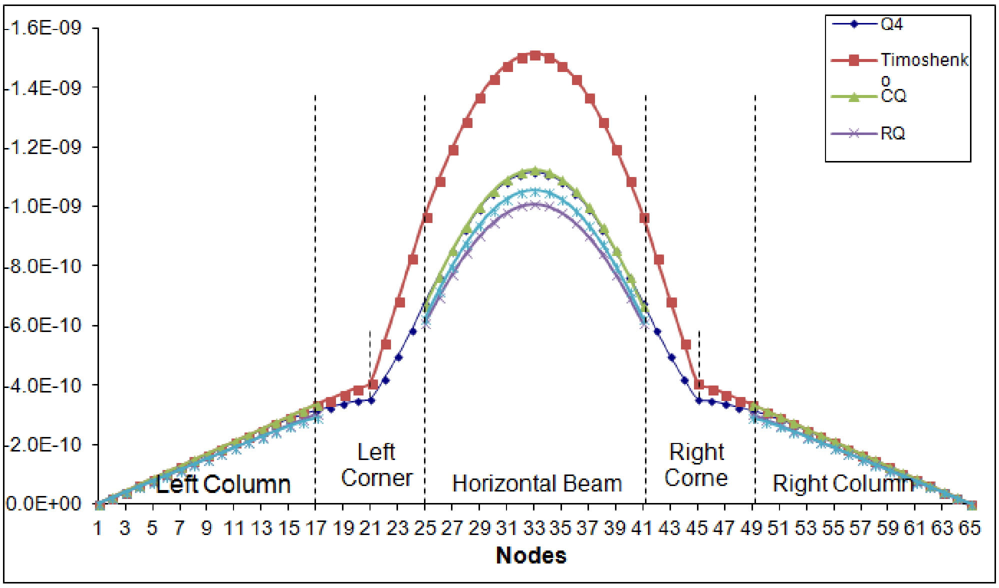


FIGURE 36: UY (m) DISPLACEMENT OF THE CENTRAL LINE NODES ALONG Y (LOAD CASE 1)

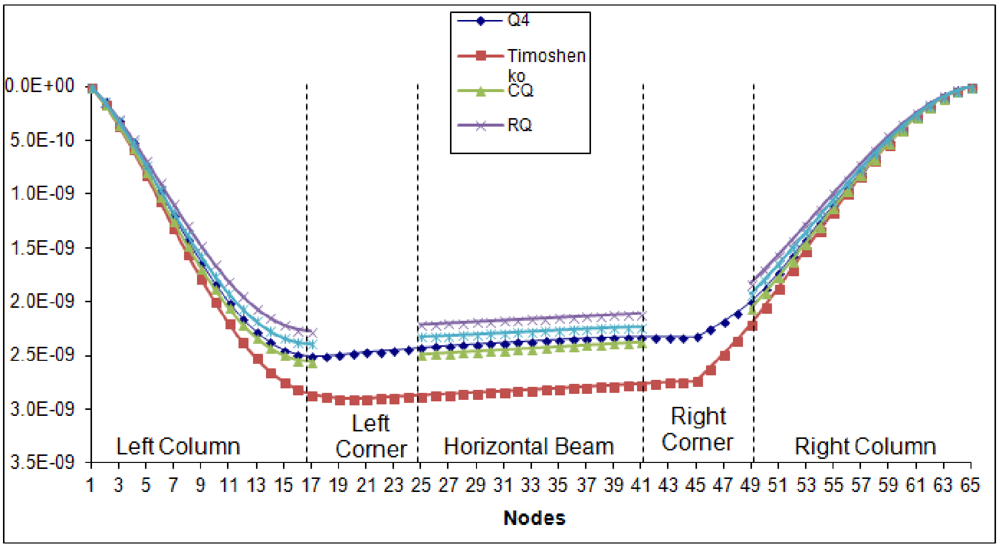


FIGURE 37: UX (m) DISPLACEMENT OF THE CENTRAL LINE NODES ALONG X (LOAD CASE 2)

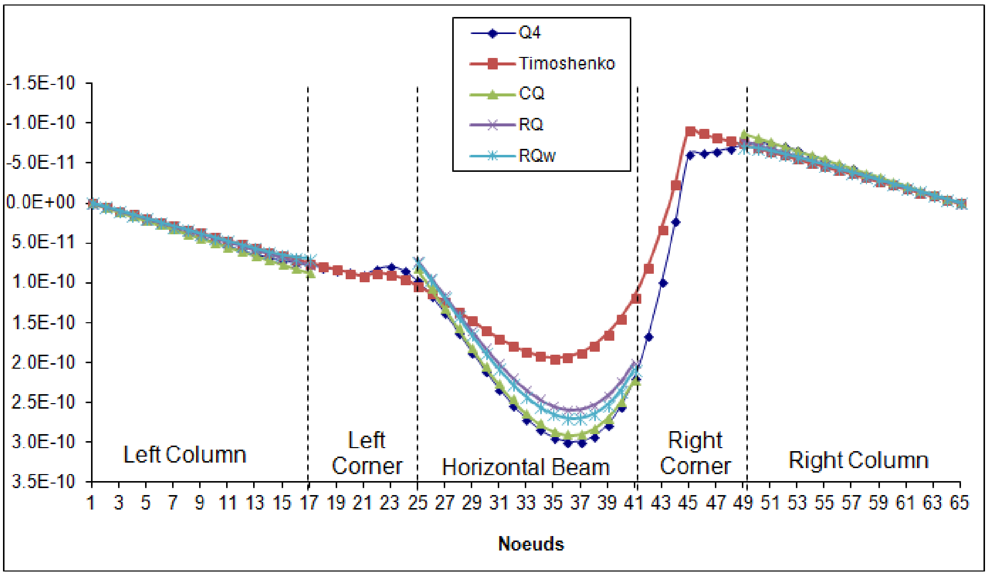


FIGURE 38: UY (m) DISPLACEMENT OF THE CENTRAL LINE NODES ALONG Y (LOAD CASE 2)

The previous graphs show that the RQw is more flexible than RQ, whereas CQ gives best results.

VIII. SUMMARY

The following table summarizes the results coming from all models:

TABLE 3
COMPARISON OF ALL MODELS AND ALL LOAD CASES

Model	Node (where E = E_{max})	Load case	Displacement	E_{max} par rapport à H8	E_{max} par rapport à Q4
H8	15 ou 51	1	UX	0%	1.55%
H8	33	1	UY	0%	1.90%
H8	16	2	UX	0%	3.01%
H8	39	2	UY	0%	2.92%
Q4	15 ou 51	1	UX	1.55%	0%
Q4	33	1	UY	1.90%	0%
Q4	16	2	UX	3.01%	0%
Q4	39	2	UY	2.92%	0%
Timoshenko	14 ou 52	1	UX	15.22%	16.28%
Timoshenko	33	1	UY	38.45%	35.87%
Timoshenko	36 ou 37	2	UX	21.48%	18.19%
Timoshenko	38 ou 39	2	UY	28.44%	30.38%
CQ	**15 ou 51**	**1**	**UX**	1.52%	**2.59%**
CQ	**17 ou 49**	**1**	**UY**	2.74%	**1.69%**
CQ	**28**	**2**	**UX**	5.52%	**2.70%**
CQ	**25**	**2**	**UY**	4.48%	**4.10%**
RQ	14 ou 52	1	UX	6.30%	7.61%
RQ	33	1	UY	7.71%	9.44%
RQ	16	2	UX	6.48%	9.19%
RQ	37	2	UY	8.72%	10.97%
RQw	14 ou 52	1	UX	4.34%	5.71%
RQw	33	1	UY	4.34%	5.37%
RQw	17	2	UX	1.66%	4.52%
RQw	36	2	UY	6.77%	7.84%

It is obvious that the CQ model gives best results when comparing with the Q4 model.

IX. CONCLUSION

Replacing the volumetric modeling of a simple frame by traditional 1-D elements leads to a weak simulation in static analysis. It is found out that the problem arises from corner connections. Modeling these connections by rigid links produces a stiff structure. Using CQ elements at the corners improves the results in the static analysis. Consequently, the number of elements and the number of DOF of the finite element model are dramatically reduced from hundreds of thousands to merely hundreds.

REFERENCES

[1] Alghamdi, S, "Dynamic stiffness matrix and load functions of Timoshenko beam using the transport matrix", Computers and Structures, 79 (2001) 1175-1185.

[2] Cook R. Concepts and applications of finite element analysis. 4th Edition. Wiley 2002.

[3] Craig R. Structural Dynamics – An introduction to computer methods. John Wiley & Sons 1981.

[4] Dohrmann CR, Key SW, Heinstein MW. "A method for connecting dissimilar finite element meshes in two dimensions". International Journal For Numerical Methods In Engineering 2000; **48 (5)**, pp. 655-678.

[5] Dohrmann CR, Key SW, Heinstein MW. "Methods for connecting dissimilar three-dimensional finite element meshes". International Journal For Numerical Methods In Engineering 2000; **47**, pp. 1057-1080.

[6] Hoit M. Computer Assisted Structural Analysis and Modeling. Prentice-Hall 1995.

[7] Kassimali A. Structural Analysis. Second edition. Brooks/Cole 1999.

[8] Kattner M, Crisinel M. "Finite element modeling of semi-rigid composite joints". Computers and Structures 2000; **78**, pp. 341-353.

[9] Quiroz L, Beckers P. "Non-conforming mesh gluing in the finite elements method". International Journal For Numerical Methods In Engineering 1995; **38 (13)**, pp. 2165-2184.

[10] Rao S. Mechanical Vibrations. 4th Edition. Pearson Education 2004.