

## **CHAPTER 4**

### **FINITE ELEMENT ANALYSIS OF COLD-FORMED STEEL BEAMS**

#### **4.1 GENERAL**

Modeling is one of the important features in Finite Element Analysis. This chapter discusses the finite element modeling of the Cold-Formed steel beams with concrete encased trapezoidally corrugated web. The finite element analysis program ANSYS is used to create the model of the tested specimens of beams. Using these models, the ultimate loads and total deformation of steel beams with plain web, trapezoidally corrugated web, concrete encased plain web and concrete encased trapezoidally corrugated web under simply supported end conditions are compared with experimental results.

#### **4.2 ABOUT ANSYS WORKBENCH**

ANSYS Workbench capabilities include a unique and extensive materials and sections library for concrete and steel structures. In addition, the user could introduce any shape or material into the corresponding ANSYS Workbench libraries. A user-friendly beam and shell postprocessor includes listing and plotting section geometry, reinforcements, beam stresses and strains inside the cross-section. The skilled combination module, selects loads and coefficients for logic code combinations. Results embrace concomitance at

element and global level as well as worst load arrangements in beam, shell and solid elements.

The analysis is carried out in three stages such as

1. Preprocessor
2. Solution
3. Postprocessor

### **4.3 MODELING OF COLD-FORMED STEEL BEAMS**

Finite Element Modeling of cold-formed steel beams in ANSYS Workbench consists of three stages, such as

- Selection of element type
- Material properties
- Modeling and meshing the geometry

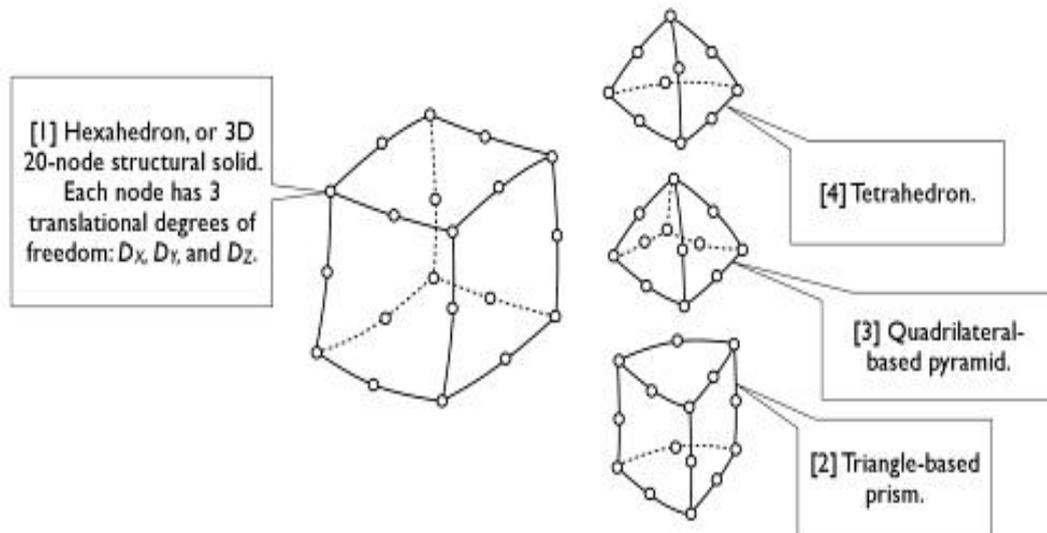
#### **4.3.1 ANSYS Workbench Element Shapes**

ANSYS literally provides hundreds of element types. ANSYS Workbench, however, directly supports only a portion of them. Other elements types can be accessed by using APDL. To identify these element types, each elements type is assigned a code name (e.g., SOLID 186, PLANE 183, etc ). In 3D cases, the available shapes include hexahedral (6-faced), tetrahedral (4-faced), triangle-based prism (5-faced) and quadrilateral-based pyramid (5-faced). By default, the ANSYS Workbench automatically chooses appropriate element types from an element library. According to the types of the structural bodies, current version of the ANSYS Workbench supports only 4 body types; 3D Solid Body, 2D Solid Body, 3D Surface Body and 3D line Body.

By default automatically 3D Solid bodies were chosen to model the cold-formed steel beams with concrete encased trapezoidally corrugated web.

#### 4.3.1.1 3D Solid Bodies

ANSYS Workbench meshes a 3D Solid body with SOLID 186, a 3D 20-node second-order structural solid element. The element is hexahedral in its most general shape. By combining some of nodes, the element can degenerate to a triangle-based prism, quadrilateral-based pyramid or tetrahedron. Figure 4.1 shows the 3D Solid Bodies.



**Figure 4.1 3D Solid Bodies**

#### 4.3.2 Material Properties

The values obtained from the tests were given as input in ANSYS as properties of steel and concrete. Coupon tests were conducted in order to determine the modulus of elasticity and yield strength of cold-formed steel and cube tests were conducted to determine the compressive strength of concrete and the values obtained are given in Table 4.1.

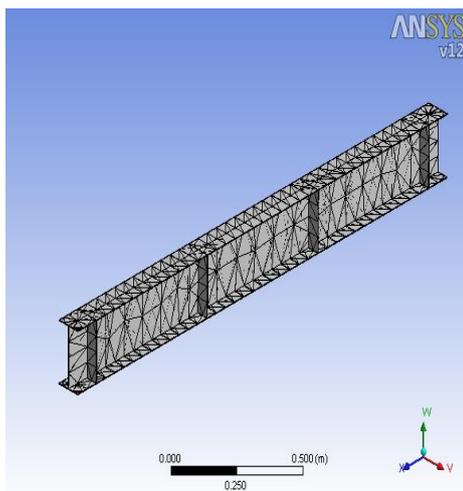
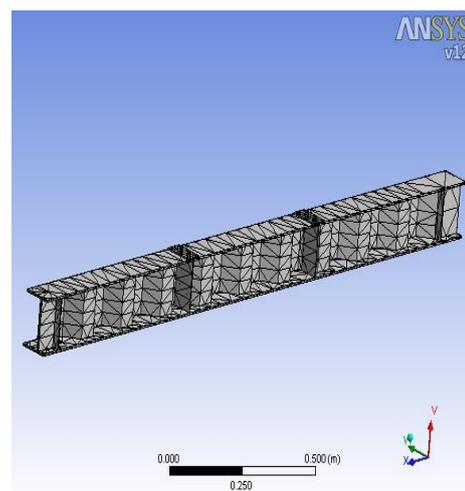
**Table 4.1 Material Properties for Cold-Formed Steel and Concrete**

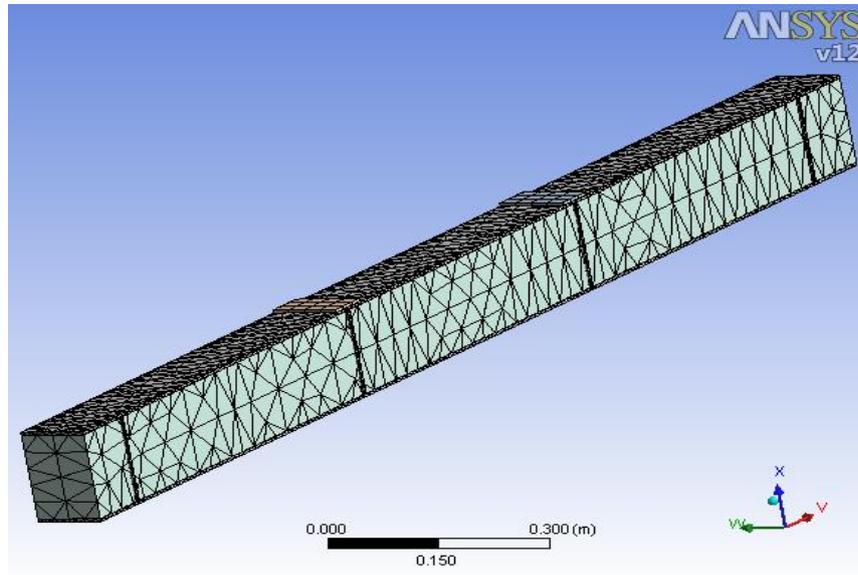
Sl. No	Description	Values
1.	Young's modulus of cold-formed steel	$2.010 \times 10^5 \text{ N/mm}^2$
2.	Yield strength of cold-formed steel	$422 \text{ N/mm}^2$
3.	Compressive strength of concrete	$35.95 \text{ N/mm}^2$

### 4.3.3 Modelling and Meshing the Geometry

After assigning appropriate material properties, the model was built in using graphical user interface. Figure 4.2 (a)-(c) shows the meshing of the beam with plain web, trapezoidally corrugated web and concrete encased trapezoidally corrugated web beams.

The important step in finite element analysis after the creation of the model is meshing. Meshing a 3D solid body with SOLID 186, a 3D 20-node second-order structural solid element. By combining some of nodes, the element tetrahedron is chosen for meshing the solid elements.

**(a) Beam with Plain Web****(b) Beam with Corrugated Web****Figure 4.2 (Continued)**

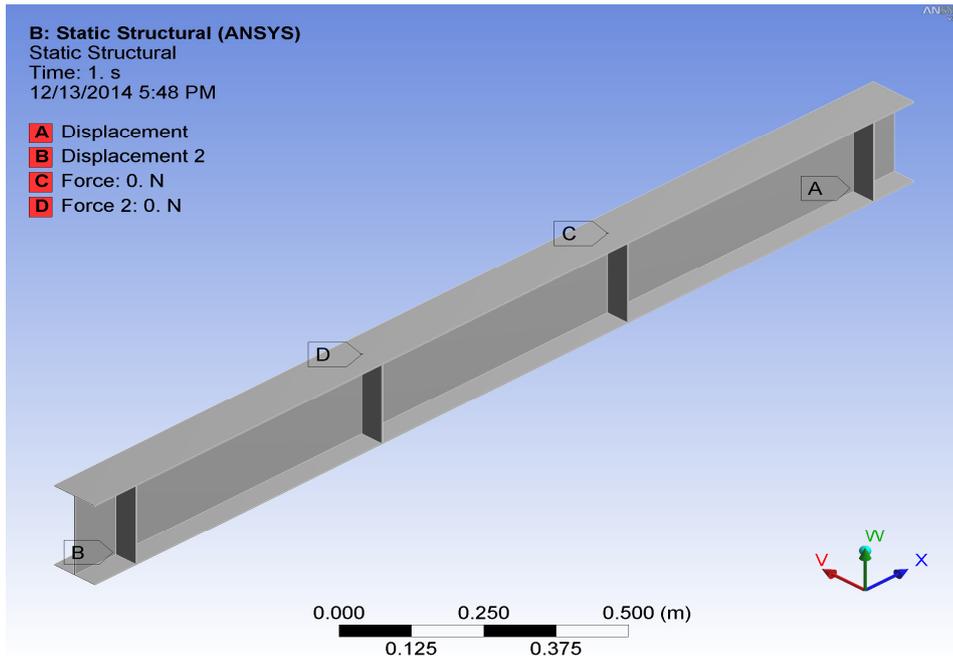


**(c) Beams with Concrete Encased Trapezoidally Corrugated Web**

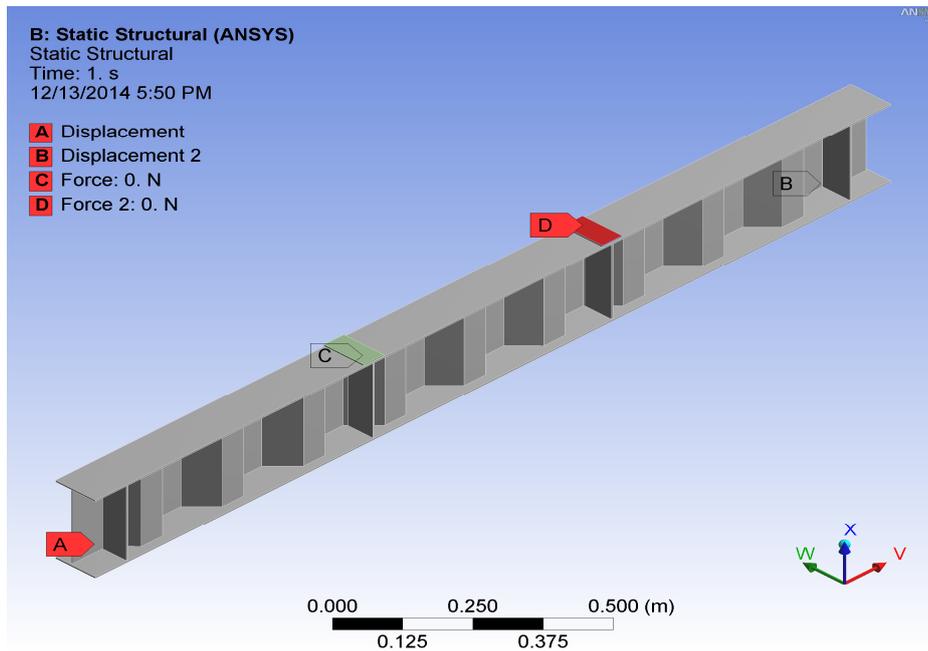
**Figure 4.2 Meshing of the Beams**

#### **4.4 APPLICATION OF LOADS AND BOUNDARY CONDITIONS**

Displacement boundary conditions are needed to constrain the model to get a unique solution. To ensure that the model acts the same way as the experimental beam the boundary conditions need to be applied at points, where the supports and loadings exist. One end of the beam was hinged and the other end was given roller boundary conditions. Loads were applied at one-third distance from the supports at a uniform rate till the ultimate failure of the specimens occurred. The beams with boundary conditions are shown in Figure 4.3(a)-(c).

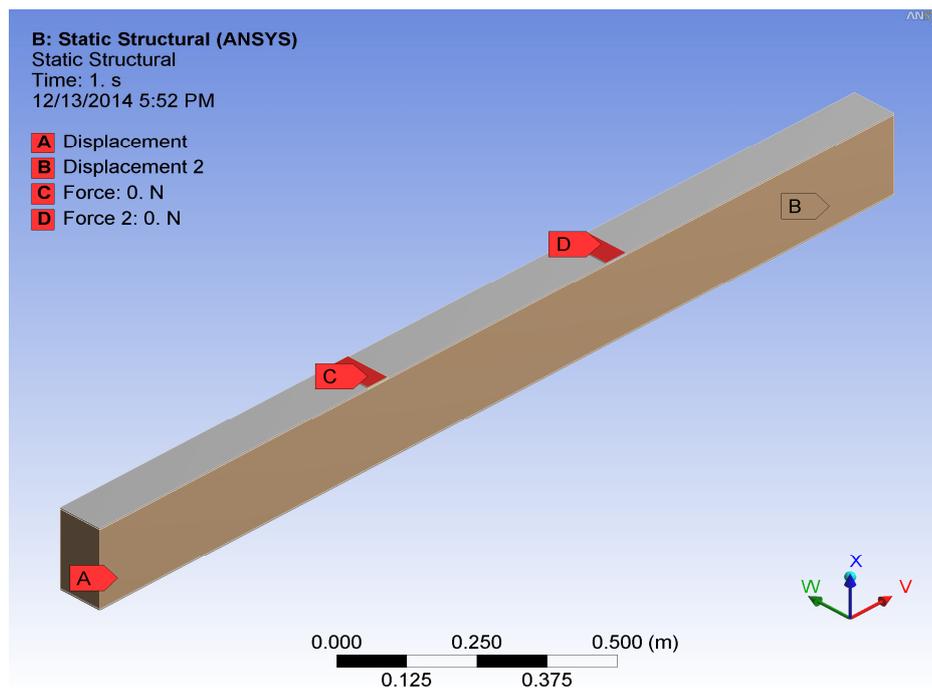


(a) Beam with Plain Web



(b) Beam with Trapezoidally Corrugated Web

Figure 4.3 (Continued)



(c) Beams with Concrete Encased Trapezoidally Corrugated Web

**Figure 4.3 Beams with Boundary Conditions**

## 4.5 SUMMARY

The cold-formed steel beams with plain web, trapezoidally corrugated web, concrete encased plain web and concrete encased trapezoidally corrugated web were modeled in ANSYS.