Numerical Analysis of CASTALLED Beam Using Ansys

Ajay Kumar Singh¹, Nikhil Garg²

¹Department of Civil Engineering, Graphic Era Deemed to be University, Dehradun, Uttarakhand India
²Department of Civil Engineering, Graphic Era Hill University, Uttarakhand India

ABSTRACT
Using ANSYS WORKBENCH 16.0 for Finite Element Analysis, this dissertation examines various failure patterns and the impact of web openings on different structural characteristics. The current study examines the static evaluation of single and double solid web, single and double web castellated beams as well as the nonlinear time historical analysis of these beams. In this study, steel beams with single, double and single solid webs, double and single web castellated beams are designed and studied using finite element analysis. This shows that castellated beams have a higher load carrying capability than their parent beam sections up to the serviceability limit (i.e. solid beams). Lateral torsional buckling is the primary mode of failure, and it is mitigated in double solid web and double web castellated beams as compared to single solid and single web castellated beams.

Keywords: Stress, Finite Element Analysis, Deflection, and Stellated Beam

INTRODUCTION
ANSYS The finite element-based package is normally used for structural analysis, fluid mechanics, heat transfer, acoustic and electromagnetic problems. The castellated beam can be modeled by a nonlinear and linear behaviour. The behaviour of the material is selected as per its load-carrying capacity, shear, and tensile strength. In a nonlinear analysis of structure either material or geometric non-linearity are involved. Due to material non-linearity, the stress-strain curve becomes nonlinear with variation in the secant modulus of elasticity. The geometrical nonlinearity is considered for bodies of irregular shape. In the present study, the finite element analysis is carried out for solid beam and castellated beams of depth 150 mm, 225 mm to determine their maximum load-carrying capacity, deflection, and stress.[6-9] A 3-D finite element (FE) model is established to replicate the behaviour of solid and CBs having the cross-section of I-shaped with FE packages ANSYS 16.0 and 19.2.

The invention process is driven by necessity. Engineers today are continually working to enhance the material qualities and design and building methods. Due to its beneficial construction purposes, such as their light weight, low cost, relatively high reaction, and ability to be quickly assembled on the construction site, castellated beam use has become increasingly widespread. [10-12] These applications make use of the castellated beams' improved strength and affordability. In addition, there are many castellated beam failure modes, including the Vierendeel collapse method, web post buckling, flexural failure, lateral torsional buckling, and shear failure. Because the double web
castellated beam is available, these modes of failure are reduced. The behaviour of a double web castellated beam as the width of the opening increases is not covered in the literature that is currently available. These days, interest in using steel for structural purposes in buildings is growing quickly. In the middle of the 1930s, an engineer working in Argentina named Geoffrey Murray Boyd made improvements to built-up structural components, one of which is the castellated beam. [13] In order to enhance the entire beam depth by 50% and enhance structural behavior against bending, castelled beams are created by flame cutting rolled beams along their centre lines and then putting the two halves together by welding. Cost of structural steel is decreased with a castellated beam.

beams with castellations or perforations on the web section. The solid beam's web half is cut in a zigzag pattern, and the two halves are then put together such that the web part has castellations. Then, it is joined by welding to create a castellated beam. With castellated beams, the thickness of the beam can be increased without adding any more steel. In this work, a single web castellated beam and a double web castellated beam with hexagonal apertures are taken into consideration. [14] To conduct finite element analysis, utilise Ansys 16.1. The spans of the beams are 6 metres, 9 metres, 12 metres, and 16 metres, with uniformly distributed loads up to the reliability limit. with various parent [15]sections for castellated beams with a single and double web. When performing nonlinear time history analysis, earthquake ground motion, specifically modified

**Finite Element Modeling**

**SHELL181 Element**

For analytical modelling in ANSYS, the thick shell structures SHELL181 element is suitable for the present study. At each node, it has a 4 -nodded element having 6-Degrees of Freedom (DOF): rotations about the x, y, z-axes, and translations in the x, y, and z directions (The element has translational degrees of freedom only if the membrane option is used). In mesh generation, the degenerate triangular option must be used as filler elements. For linear, large rotation, and/or large strain nonlinear applications SHELL181 is compatible. In nonlinear analyses change in shell thickness is considered. Both full and reduced integration schemes are supported in the element domain. SHELL181 considers follower effects of distributed pressures (load, stiffness). Figure 5.1 shows a detailed view of SHELL181element.
Reinforced concrete

The beam was modeled using three-dimensional SOLID65 elements. This element is specifically applicable to concrete as it shows cracking behaviour at a specified tensile value and also crush at specified compressive strength of the concrete. It is also having the ability to take shear depending on the coefficient values given on the condition of the developed flexural cracks. The rebars mesh is formed as a link element distributed at the bottom of the beam at nodes. The rebars are modeled to find reinforcement behaviour. This element is of eight nodes. Each node has three degrees of freedom i.e. translations in x, y, and z-direction. Concrete is allowed to crack in three orthogonal directions, with crushing, creep, and plastic deformation. Tensile and compressive forces are allowed on rebars. Also, rebars undergo plastic deformation and creep. Figure 5.2 shows a detailed view of the SOLID65 element.

![Figure 5.2 SOLID 65 element](http://www.webology.org)

Steel reinforcement

Steel reinforcement is modeled as a Beam188 element as shown in Figure 5.3. This element is grounded on the theory of shear deformation Timoshenko assuming constant transverse-shear strain through the cross-section. This acts as two noded elements in 3-D and may be linear, quadratic, and cubic. Both of the nodes have 6 DOF with rotation and translation both in three mutually perpendicular axes. It is normally used in linear, rotational, or nonlinear strain applications. The 100% bonding of steel and concrete is considered.
Steel plates
These elements are named SOLID45 acting at support in the form of steel plates. It is composed of eight nodes. Each node has three degrees of freedom viz. translation in x, y, and z-direction. This element provides better resistance in plasticity, deflection, creep, stress, strain, etc. It provides control and reduced integration. For modeling in 3-D of solid structural elements, it is used. Refer to Figure 5.4 for the details of the SOLID45 element.
Laminates
Modeling of sandwich structures is done using the SHELL91 element by applying layers of the element on the surface of the component. This element is having 6 DOF per node allowing for translation and rotation both in three perpendicular directions. It is suitable for three layers of laminate. For more than three layers, SHELL99 is used which takes less time as depicted in Figure 5.5.

ANSYS modeling using mechanical APDL
The SOLID 185 element is utilised as indicated in Figure 5.6 for analytical modelling in ANSYS utilising mechanical APDL and for preparing solid structures 3-D modelling. It is clearly defined by its eight nodes, each of which has three degrees of freedom (DOF), or translation along x, y, and z. The element is capable of large strain, big deflection, yield stress, creep, and hyperelasticity. Additionally, it offers the capacity to use mixed formulations to simulate the distortion of fully incompressible hyperelastic elements as well as virtually incompressible elastoplastic elements. It comes in two formats: 1. uniform structural solid 2. A structural solid with layers. When employed in irregular locations, it allows for tetrahedral, prism, and pyramid degenerations. Different element technologies, like increased strains and B-bar universally reduced integration, are offered.
Figure 5.6 Homogeneous structural solid geometry of SOLID 185 element

Properties of material
For modeling of the CBs, linear and nonlinear analysis is considered. After defining geometry and boundary conditions, the material properties are assigned to start pre-processing. Various elements used in the modeling and bilinear properties of materials are shown in Table 5.1.

Table 5.1 Elements used in the modeling and properties of the material.

<table>
<thead>
<tr>
<th>Materials</th>
<th>$\rho$</th>
<th>$E$</th>
<th>$\gamma$</th>
<th>$f_y$</th>
<th>Used Element</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel</td>
<td>7850</td>
<td>2.1X10$^5$</td>
<td>0.3</td>
<td>250</td>
<td>Shell 181</td>
</tr>
<tr>
<td>Steel</td>
<td>7850</td>
<td>2.1X10$^5$</td>
<td>0.3</td>
<td>250</td>
<td>SOLID 185</td>
</tr>
</tbody>
</table>

Loading conditions and geometry
From the overall length of 1700 mm, an effective length of 1600 mm simply supported beam is considered. The cross-sectional dimension of the beam is 80 mm $\times$ 225 mm. Figure 5.7 shows the castellated model of the sinusoidal opening having fillet radius 1/4th of hexagonal opening. The analysis end conditions applied are simply supported at the initial stage followed by pure bending under application of two-point loads at L/3 distance from both ends. During experimentation, two points loading is applied at the main beam which is 533.33 mm from both ends face of the beam. This two-point loading is used to give the deflection at the center of the main beam. In the modeling, two equal loads are placed at a distance of L/3 from the beam support. Mesh size is taken 1 mm throughout, to get results accurately. Figure 4.5 shows an experimental fabricated beam model of the sinusoidal castellated beam. The ANSYS beams are modeled with various opening configurations such as fillet radii, angle of opening, and equal length of opening for rectangular shapes. In the analytical investigation, the experimental ultimate load is applied over the prepared model of ANSYS. Finally, the deflection is noted for a corresponding ultimate load, curvature, and bending moment is workout. After preparing the steel beam model over ANSYS, the next step is to do the finer meshing which is reflected in Figure 5.8. That shows a form of meshing as triangular, square, hexagonal, or octagonal. The two-point load has placed on the beam so that pure bending is formed. The ends of the beam are simply supported. The ultimate load obtained in the experimental program will be assigned to the beam model. The maximum deflection generated in castellated beams is shown in Figure 5.9. The Von-misses stresses and maximum shear stares are generated in the beam throughout the loading as per Figure 5.10 and Figure 5.11.
Figure 5.7 Position of supports and loads for CSBIS1

Figure 5.8 Typical finite element mesh for CSBIS1

Figure 5.9 Maximum deflection in CSBIS1
Conclusion
All the castellated beams are modeled with finite element modeling using ANSYS. Mesh size of 1 mm with a standard deviation of 0.17 mm is considered for ANSYS modeling. Two identical loads are placed at L/3 distance from both end face of the beam because of pure bending consideration. At both simple supports of the castellated beam, rotations (θy and θz), as well as the displacements (dx, dy, and dz), are restrained and rotation (θx) is permitted. In this study, the investigation of the experiment is conducted up to yield, maximum or breaking point. Therefore elastoplastic (bilinear stress-strain relationship) is taken into consideration for the analytical investigation of ANSYS.

Reference: