ABSTRACT

The bracket is an architectural element of structural or decorative member. It is attached to the walls for supporting the structures. In ancient architecture, brackets are used for decorating the structures and made with stones. It can be made of wood, stone, plaster and metals etc. and now a day's concrete structures are using. Bracket is a 3D-model structure used to support the architectural items like balcony, arches, roofs, parapets and shelfs, etc. Brackets are used in aerospace industry and in automobile industries. In the current technology concrete structures used for the construction of brackets. It is used to carry the weights of structure to the walls and to strengthen at an angle. Mainly there are two types of brackets corbel and console. In present study we are using corbel bracket. ANSYS is a software which is used to analyse the model. In this we are going to analyse a 3D- model bracket for that we are going to use the 185 & 187 element types and we are using linear static analysis method to analyse the structure by giving the material properties like young's modulus and poison's ratio and load conditions and element properties. And we can analyse the bracket after giving all the values in the tool. by this analysis we can get the accurate results.

Keywords: Bracket, ANSYS, linear static, corbel, console