Modelling framework joints in ANSYS software complex

N A Buzalo, A S Alexeeva and B A Chernykhovsky
Department of Urban Planning, Design of Buildings and Structures, Platov South-Russian State Polytechnic University (NPI), 132, Prosvesheniya st., Novocherkassk 346400, Russia
E-mail: buzalo_n@mail.ru

Abstract. Modern methods of building design are largely based on the creation and analysis of a 3D model of a structure in a specialized CAE complex using the finite element method. The article describes the problem of designing the junctions of crossbars and columns in the metal frame of high-rise buildings. A computational model of a flange joint is created in the form of a solid three-dimensional model in the ANSYS software package, a visual display of equivalent stress fields was obtained, and the calculation results are analyzed. The method of strength analysis of junction points of frame systems using CAE systems makes it possible to more effectively take into account many options in the design due to the system of parametric values, and also analyze in more detail the stress-strain state of the calculated structure.

1. Introduction

Special attention should be paid to the calculation and design of the joints of the main bearing elements in frame building designing. Modeling the framework with CAD-systems is used for the last twenty to thirty years, while creating models of framework connections using CAE-systems has been used in designing relatively recently. The main purpose of this paper is to create a model of the connection, the study of its stress-strain state, the identification of "weak" places of the connection with the development of local plastic deformations.

For the first time, the term “frame building” began to be used in the construction of high-rise building, when steel and reinforced concrete core elements (racks, beams) replaced the massive load bearing brick walls. The main advantage of such systems is that the functions of the structural elements are clearly divided into bearing and non-bearing. The framework is a spatial-pivotal system that perceives all loads directed at it arbitrarily and transfers them to the foundation. Flat stiffening diaphragm provides longitudinal and transverse rigidity, as well as torsional rigidity. Torque relative to the vertical axis laid out on the tangential forces in the planes of the diaphragm walls. A variety of placement of stiffness diaphragms on the floor plan is used. Vertical supporting structures of frame buildings practically do not limit layout options. The vertical supporting structures of frame buildings do not limit the floor planning options. This explains the widespread use of frame systems in the construction of buildings for various purposes - administrative, commercial, residential, etc.

Using modern approaches to the design of steel frames of buildings and structures, the designer develops a spatial model consisting of rods rigidly or hingedly connected, and perceiving different types of effects - tension, compression, bending and torsion. The use of different types of joining elements significantly affects the stress-strain state of the whole structure. This article discusses the
structures that are part of the spatial framework of the building, with two possible connection options - rigid and hinged, according to the unified design solutions proposed by the 2.440-2 series of 1989.

Joint-type coupling eliminates the appearance of a bending moment in the constraint zone, but releases the degree of freedom of rotation. Therefore, this type of connection is often not used in the design of structural components necessary to ensure the spatial rigidity of the system (for example, the mating of the column with the foundation). In practice, such joints, which slightly prevent rotation of the beam on the support, are used for horizontal framework elements, located in the longitudinal direction, in most beam structures of floors and coverings. The “hinge effect” is ensured by the fact that the beam fastening is performed through a plate, an L-bar or a flange with bolts located sufficiently close to the beam axis (Figure 1). In some cases, the beam may also be supported by a lower shelf on a supporting bracket made of an L-bar.

Figure 1. Connection of beam and column.

2. Materials and methods

2.1. The main features

The rigid connection of the beam with the column, which is characteristic of the frameworks of high-rise buildings (Figure 2), is mainly ensured by fixing the beam to the column through the flanges. The number of bolts concentrated in the upper part of the flange prevents rotation of the end of the beam.

Figure 2. Steel frame scheme.

The rigid connection of elements eliminates any movement and rotation of the support section, as a result of which a bending moment occurs in the constraint zone - the moment of support reaction. Bending moment, like other types of internal forces acting in a body in a stress-strain state, respectively, creates internal stress. This stress must not exceed the ultimate limit stress of the material of which the calculated structural element consists.

In addition to the calculation of core structural elements, in the junction assembly it is required to perform a calculation of structural elements providing the intended type of joint (plates, ribs, flanges,
supporting brackets, bolts, etc.) and welds (Figure 3). In this case, strength calculations are required for shearing and crushing of the metal.

![Steel frame scheme.](image)

Figure 3. Steel frame scheme.

It should be noted that in fact there are no perfect hinges and rigid connections in building structures. This is a sufficient convention. The friction force is ignored and it is assumed that along the Y axis the movement of the beam is not limited by anything. It is also important to avoid incomplete clamping (when there is no construction with small efforts to turn the structure, but as the force increases, the connection is broken and the rotation occurs). Or, the rigidity of the node supporting the beam on the column in the frame was provided by welds, but the weld is destroyed. The beam continues to lean on the column, but can already turn on a support. In this case, the curve of the bending moments changes significantly - the moments on the supports tend to zero, but in the middle of the beam the moment of forces increases. The beam section was calculated according to the design scheme with rigid joints and is not ready for perception of the increased moment, which can lead to destruction. Therefore, a rigid connection should always be designed for the maximum possible load.

2.2. Stages of strength calculation

For the strength calculation of the connection of the frame structure using the finite element method in ANSYS, a calculation of the statically indefinable frame in the SCAD software complex was carried out and the optimal cross sections for the elements were determined (the junction point of the bolt B-1 to the column K-1 through the flange using 8 high-strength M27 bolts, figure 3). As a result of the calculation, it was found out that the most loaded fragments of the system are node 14 and the elements adjacent to it. To create a computational model in the ANSYS software package, the results of the calculation of a statically indefinable frame (Figure 2) in the SCAD software package, namely, translations in a section located at a distance of two section heights from the junction, were used. These translations are applied to the calculation model in ANSYS in the initial boundary conditions corresponding to the analysis case.

The model of the calculated junction unit is made in the form of a solid three-dimensional model (Figure 4). Welds are separate geometric body. This decision was made to simplify the process of creating a finite element model. Column and beams are limited in length to three section heights. This is also done to simplify the calculation model, reducing the required resources and reduce calculation time.
This design model is symmetric in two planes, therefore, to simplify the task, it can be simplified (Figure 5). For the calculation, it is enough to use a quarter of the design model, since the planes of symmetry take into account the behavior of the complete structure, but at the same time the required computer resource is reduced by four times.

The next step in the structural strength analysis is to define the contact surfaces. Two types of contacts were used to simulate contact interaction - bonded between the body of the seam and the elements to be joined and frictional between the elements of the bolted joint package. The main characteristic of frictional contact is the coefficient of friction (in this case, the coefficient of friction of steel on steel). The value of the friction coefficient is selected according to table 6.7 [3]. For calculation, it is enough to choose the average value in the range indicated in the table, equal to 0.17.

![Figure 4. Computing model.](image1)

![Figure 5. Simplified computing model.](image2)

The bonded contact is set between the welding body and the elements being joined (Figure 6). The frictional contact is defined between the contacting faces of the elements, as shown in Figure 7.

![Figure 6. Contact surfaces – bonded.](image3)

![Figure 7. Surface contacts – frictional.](image4)

After specifying the contacts finite elements mesh follows. The integral elements of the bolt and columns are divided into parts are shown in Fig.4. This is done for easy mesh. In the zone where the
cross beam is a continuous elongated cross section regular hexahedra, closed to the ideal form, are used. However, in the zone of surface contacts such an “ideal” forms are impossible, as a result it is necessary to create a lower-quality mesh. Separating solid structural elements into components helps to achieve the greatest number of correct quality elements of a finite element mesh. The mesh of finite elements ready for calculation is shown in Figure 8.

![Figure 8. Model finite element mesh.](image)

The final stage of preparation of the computing model is the imposition of boundary conditions of sealing and loading. In this case, the stress-strain state of the connected elements is achieved by pretension of the bolts, as well as forced removals, the values of which are obtained from the calculation of the frame by the SCAD. Sets the offset of the lower and upper sections of the column, the shift of the cross section of the crossbeam down the z-axis (Fig. 9). The characteristics of the steel are taken in accordance with the basic physical and mechanical properties of steel C345 [9].

![Figure 9. Model finite element mesh.](image)
Next, the process of analyzing the above-described scheme starts, as a result the visual display of the equivalent stress fields is obtained and shown in Figure 10.

**Figure 10.** Stress-strain state in connection.

3. Results

As a result of the analysis, a peak stress value of 392 MPa (which exceeds the yield stress of steel C345) was found in the zone of connection of the bolt with the flange, namely in the lower horizontal weld (Figure 11). This confirms the pattern emphasized by E.I. Belenya [11] on the violation of the Zhuravsky law on shear stress distribution along the height of the supporting cross-section of the beam and the possibility of local plastic deformations. The stress concentration is fixed at the bottom of the web, and the degree of concentration depends on the ratio of the web thickness and the cross-sectional area of flange. The lower horizontal weld has more constructive meaning and does not significantly affect the bearing capacity of the connection, but due to the fact of the possible progressive collapse, measures should be taken to strengthen the connection in the identified weak spot.
Figure 11. Peak stress value in weld.

The method of strength analysis of frame systems using the CAE-system of ANSYS PC allows to take the variability in the design due to the system of parametric values. The use of this system allows more clearly and detailed show the stress-strain state of the calculated structure.

References
[1] SP 20.13330.2016 Loads and impacts p 95
[2] SP 16.13330.2017 Steel structures p 147
[3] GOST 27751-2014 Reliability for Constructions and Foundations. General principles
[4] Sagdeev Y A, Kopysov S P, Novikov A K 2011 Introduction to FEM p 44
[5] Ovcharenko V 2004 Solution of Engineering Problems Using FEM, DSMA
[6] Basov K A 2009 ANSYS for Designers (Moscow: DMK Press)
[7] Agapov V P 2004 FEM in Statics, Dynamics and Stability of Structures, 2nd ed. (Moscow: DIA)
[8] Alifutov N 2000 Stability of Elastic Structures (New York)
[9] GOST 27772-88 Rolled Products for Structural Steel Constructions. General specifications
[10] Chernavsky S A, Reschikov V F (eds.) 1976 Metalist T.1. Ed.3rd. Mechanical engineering p 768
[11] Melnikov N P (ed.) 1980 Steel structures. Designer's Handbook. 2nd ed. (Moscow: Stroyizdat) p 776
[12] Kudishin Yu I (ed.) 2007 Metal structures (Moscow) p 688
[13] SNIP II-23-81 1989 Manual on the Design of Steel Structures (Moscow: TsITP) p 149
[14] Pisarenko G S, Yakovlev A P, Matveev V V 1988 Handbook of Resistance Materials 2nd ed., (Kiev: Naukova Dumka) p 736
[15] Logvinov V B, Alekseev S A 2009 Mechanics of Materials and Designs. Selected lectures: textbook, SRSPU (NPI) (Novocherkassk) p 84
[16] Nakasone Y, Yoshimoto S, Stolarski T A 2006 Engineering Analysis with ANSYS Software (Butterworth-Heinemann) p 480
[17] Basov K A 2005 ANSYS. User reference Text (Moscow: DMK Press) p 640
[18] Birger I A, Panovko I G 1968 Strength, Stability, Oscillations. Handbook. Volumes 1, 2. (Moscow: Mechanical Engineering) p 567
[19] Bleich F 1959 Buckling Strength of Metal Structures (Fizmatgiz) p 544
[20] Balena E I 1986 Metal structures (Moscow) p 560