Results of Studying the Yield Strength of the Spatial Structure Joint Using CAE Systems

Vadim Alpatov

1Samara State Technical University, Academy of Architecture and Civil Engineering, Molodogvardeyskaya St., 194, Samara, 443001, Russia

avu75@mail.ru

Abstract. Numerical experiments of a spatial structure joint are presented. Studied the spatial structure joint created by the authors. The joint of spatial structure is protected by a patent of the Russian Federation. The studies were performed in four software systems: Lira (DNDIASB, Kiev), SCAD (SCAD Group, Kiev), CosmosWorks (Structural Research & Analysis Corp., Los Angeles), ANSYS Design Space (ANSYS, USA). Calculations purpose – evaluate the performance of each settlement complex. The evaluation of each computational complex was carried out according to the model creation time, computation time, FE mesh quality and ease of use of the program complex. The results of the evaluation of the accuracy and time of creation of the FE mesh, the accuracy and time of solving the problem were obtained.

1. Introduction
Stress-strain analysis of spatial structure joint can be made using finite element method. For example, using programs, such as: Lira (DNDIASB, Kiev), SCAD (SCAD Group, Kiev), CosmosWorks (Structural Research & Analysis Corp., Los Angeles), ANSYS Design Space (ANSYS, USA) and others.

Important to know abilities of every calculation complex. It is usable to know how comfortable using each complex is. Important to know result accuracy. Also usable to get information, which calculation complex is the most convenient for spatial construction joint research. Calculation complex effectiveness rating will be made using following criteria:

1. Time, spent on calculation model creation.
2. Quality of created calculation model.
3. Time consumption of calculation.
4. Calculation result accuracy.
5. Convenience of using program.

2. Materials and methods
Nodal connector, made by authors, was used for numerical experiment (figure 1) [1-3]. That joint was secured by Russian Federation patents [4, 5]. Joint has quite complicated geometrical shape.
Calculation models of joint for analysis in Lira and SCAD were fully made by those complexes resources. Calculation model geometry for analysis in CosmosWorks and DesignSpace were built in SolidWorks graphical environment. Due to noticeable difference of program complexes used in numerical experiment and some difference in their results, following is description of joint model building by program resources.

3. Creation spatial structure joint calculation model using Lira and SCAD programs
Calculation complexes Lira and SCAD have similar structure [6-9]. They are very similar. They have some difference in GUI and some result visualization abilities. Complexes use source data files of the same structure and have the same final element bases. The calculation model creation in these complexes with graphical mode of preparing initial data has some minor differences. These difference influence only convenience of using program. The user’s choice is influenced only by experience of working with the complex. There is possibility of full import/export source data files between complexes. Model is exported without data leak. This allows to use created once calculation model of joint in both complexes. Coincidence of models is 100%. Calculation model of spatial structure joint was created using the Lira program and imported it into the SCAD program.

The calculation model geometry construction is one of the main most crucial and complicated stages in the creation of the computational model. Result accuracy depends on how similar are model and physical prototype geometry. Building calculation model geometry in Lira and SCAD are made by own resources of programs. Geometry is built by assembly from finite elements (FE). FE have a defined property - geometry. FE can be line, plate, cube, prism and etc. Final joint model geometry is assembled by connection FE to each other. All FE properties are stored in information base of complexes. For this task in FE libraries are contained FE N34 and N36 (figure 2).
Figure 2. 3D FE of the programs Lira and SCAD.

a. – FE N34 (six-node element of arbitrary shape); b. – FE №36 (eight-node arbitrary shape)

Nodal connector complicated geometry description is a very labor-consumptive process. Calculation model creation process using Lira took authors a week of intensive work. In final calculation model there are 14435 nodes and 7733 elements. Final calculation model for Lira and SCAD has a number of defects:

1. Geometry of model describes physical prototype not very accurate.
2. In calculation model nearly no possibility to describe weakening by holes, strengthening by weld, edge rounding.
3. Model support is possible only through its nodes.
4. Load on the FE plane is set only in local coordinate system. No possibility to set load in global coordinate system.

4. Creation spatial structure joint geometrical model using SolidWorks program

Structural construction joint connection geometrical model for CosmosWorks and DesignSpace can be imported from program SolidWorks. It is very comfortable, because SolidWorks is well adapted for building 3D geometric objects. This geometric model accurately describes geometry of physical prototype. In geometrical model may to described weakening by holes, strengthening by weld, edge rounding. Geometric constructions accuracy is $10^{-6}$ mm. It doesn’t take much time to build a geometric model for experienced SolidWorks user. It took authors 8 hours to create geometrical model of joint.

The result of the work of the AA software complex is presented in figure 1.

5. Creation spatial structure joint calculation model using CosmosWorks program

Finite elements mesh in CosmosWorks is built automatically for object, created in SolidWorks. User is ab FE to control mesh built quality for finite elements. There is possibility to set maximum FE dimension; possibility to smaller scale mesh in local parts; possibility to add control points of create FE mesh. As a result of constructing a finite element mesh, the entire geometric model is divided into 3D FE. The program has a choice of two types of FE – four-node linear (figure 3.a) and ten-node parabolic (figure 3.b). The difference between them as an approximation of a geometric model. The use of parabolic FE improves the description of curvilinear geometry, but leads to an increase in the dimension of the problem and the time to obtain a solution. However, the results of the solution are more accurate than when using linear FE [1-3].
Authors used the linear elements (figure 3.a) for creating the spatial structure joint calculation model. Achieving strict adherence to the number of nodes and FE in the calculation models Lira and CosmosWorks is impossible because of the differences in the types of FE in these programs. Authors made efforts to approximate the number of FE in calculation models. The final calculation model for calculation using CosmosWorks program has 9491 nodes and 29152 elements.

Program allows to model supports of geometrical model in nodes, edges, plates (figure 4).

**Figure 3.** 3D FE of the CosmosWorks program.

a. – Linear Element; b. – Parabolic Element

**Figure 4.** Calculation model in CosmosWorks program

Two variants of nodal connector calculation were made in CosmosWorks program. Difference between them is quality of the elements mesh. For the second variant, the number of nodes in calculation
model is 23522, and number of elements - 75408. Such mesh of finite elements is has minimum factor scale for this task in the CosmosWorks program.

6. Creation spatial structure joint calculation model using Design Space program

Building calculation model in DesignSpace is similar to CosmosWorks. Types of FE are the same (figure 3). In DesignSpace and CosmosWorks final element meshes building procedures, setting loads and geometric model connections are the same. In DesignSpace and CosmosWorks FE mesh building algorithms and task solution search algorithms are different. Due to this there is difference in creating FE mesh time and task solution time.

It is hardly possible to get the same number of FE in DesignSpace and CosmosWorks calculation models. Due to this, in our experiment there is difference in nodes and elements number between calculation models. Author made efforts to approximate number of FE in calculation models. It was noticed, that DesignSpace to create larger scale mesh, than CosmosWorks.

Total number of nodes in joint connector calculation model is 16043, total number of elements in calculation model is 7725. Program allows to model supports of geometrical model in to nodes, edges, plates.

7. Results and discussions

Nodal connector calculations are made in these four program complexes. Calculations purpose – determine maximum stress and strain in the model. Yield stress criterion are defined by von Mises energy theory.

Von Mises yield criterion describe conditions of metals [10-12] very accurately. Von Mises Equations:

\[ VON = \sqrt{\left(\sigma_{11} - \sigma_{22}\right)^2 + \left(\sigma_{22} - \sigma_{33}\right)^2 + \left(\sigma_{33} - \sigma_{11}\right)^2 + 6 \left(\sigma_{12}^2 + \sigma_{23}^2 + \sigma_{31}^2\right)} \]  

(1)

where \( \sigma_{ij} \) – the components of the stress deviator tensor.

Task solutions, using described program complexes, are shown in table 1.

| Table 1. Task solution |
|------------------------|
| Name of settlement complex | Time spent to create geometrical model, hour | create FE mesh, min | Number of elements of calculation model | Number of nodes of calculation model | degrees of freedom in calculation model | Calculation time, min | Maximum VON, MPa |
| Lira | 60 | -/- | 7733 | 14435 | 43305 | 43 | 290.0 |
| SCAD | 60 | -/- | 7733 | 14435 | 43305 | 47 | 324.3 |
| CosmosWorks | 5 | 3 | 29152 | 491 | 28473 | 2 | 321.4 |
| CosmosWorks | 5 | 12 | 75408 | 23522 | 70566 | 23 | 295.5 |
| DesignSpace | 5 | 2 | 7725 | 16043 | 48129 | 6 | 299.4 |

8. Conclusions:

1. All four used calculation complexes give approximately same results.
2. Most effective (less labour and time consumptive for creating model and its calculation) are CosmosWorks and DesignSpace.
3. Maximum difference in tensions, defined by different complexes is less than 11%.
4. The influence of size of FE mesh on tensions values in calculation model is less than 8%.
5. Calculation model quality, created in CosmosWorks and DesignSpace, is much higher, then calculation model quality, created in Lira and SCAD.
References

[1] V. Alpatov, Numerical Studies of the Nodal Connections of Metal Spatial Frames, *IOP Conf. Series: Materials Science and Engineering*, 463, pp. 032103, 2018.

[2] V. Alpatov, A., Lukin, et al, Computer modeling and numerical studies of nodal joints of frame structures, *Urban construction and architecture*, 4(25), pp. 19-22, 2016.

[3] M. Balzannikov, V. Alpatov, et al, Usage of spatial lattice metal structures as roofing for mechanical equipment rooms of hydroelectric power stations, *MATEC Web Conf.*, vol. 73, pp. 01012, 2016.

[4] V. Alpatov, Space-frame structures, *The patent for invention RUS 2213185 (Moscow)*, 2003.

[5] V. Alpatov, The core structural design of the long size of the elements, *The patent for invention RUS 33381 (Moscow)*, 2003.

[6] A. Perelmutter and V. Yurchenko raschete prostranstvennyih konstruktsiy iz tonkostennih sterzhney otkryitogo profily, *Construction mechanics and design of structures*, 6, pp. 18-25, 2012.

[7] V. Karpilovskyi, E. Kryksunov, et al, SCAD Office. Version 21, System SCAD++, *SCAD Soft*, 2018.

[8] S. Fialko, Parallel finite element solver for multi-core computers, *Computer Science and Information Systems (FedCSIS) 2012 Federated Conference*, pp. 525-532, 2012.

[9] S. Fialko, Application of rigid links in structural design models, *International Journal for Computational Civil and Structural Engineering*, vol. 13, Issue 3, pp. 119-137, 2017.

[10] F. Beer and R. Johnston, *Mechanics of materials*, McGraw-Hill, 1981.

[11] R. Hibbeler, *Mechanics of Materials*, Second edition, Prentice Hall, 1994.

[12] J. Shigley and C. Mischke, *Mechanical Engineering Design*, Fifth edition, McGraw-Hill, 1989.