

Available online at [www.sciencedirect.com](http://www.sciencedirect.com)**ScienceDirect**

Procedia Engineering 153 (2016) 915 – 919

**Procedia  
Engineering**[www.elsevier.com/locate/procedia](http://www.elsevier.com/locate/procedia)

XXV Polish – Russian – Slovak Seminar “Theoretical Foundation of Civil Engineering”

## The Organization of Autodesk Revit Software Interaction with Applications for Structural Analysis

Sergey P. Zotkin<sup>a,\*</sup>, Elena V. Ignatova<sup>a</sup>, Irina A. Zotkina<sup>a</sup><sup>a</sup>Moscow State University of Civil Engineering (National Research University), 26 Yaroslavskoye Shosse, Moscow, 129337, Russia

### Abstract

The article deals with the export procedure of building construction parameters from Autodesk Revit to the program of structural analysis. At first, the procedure uses the C# plug-in, which extracts parameters of the construction elements from the Revit document and outputs them to a text file. The parameters can be physical (properties of materials), geometrical (size, coordinates, the cross-sectional shape) and other. Then, a special program-converter reads the file, generates a beam frame model and stores the information in a new format, which is compatible with the program of structural analysis earlier developed by the authors. This program allows to add both vertical or horizontal loads and moment to any of the beam frame nodes. The finite element method is used for structural analysis. The deformation schema is showed as the result of finite element analysis under the loads. The opportunity to get Microsoft Word document, which contains main characteristics of beam construction, is implemented.

© 2016 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license

(<http://creativecommons.org/licenses/by-nc-nd/4.0/>).

Peer-review under responsibility of the organizing committee of the XXV Polish – Russian – Slovak Seminar “Theoretical Foundation of Civil Engineering”.

**Keywords:** Autodesk Revit; C# plug-in; converter program; finite element method; finite element analysis; beam system; software.

### 1. Introduction

The rapid development of information technologies leads to new approaches to the design of buildings and building structures. Nowadays more attention is given to building information modeling (BIM) rather than CAD

\* Corresponding author. Tel.: +7-916-868-3448

E-mail address: [ZotkinSP@mgsu.ru](mailto:ZotkinSP@mgsu.ru)

design method. Building information modeling (BIM) is the process of creating, collecting and using information about a building throughout its life cycle. Unlike CAD design, information modeling is not only a graphical representation, but also information about the building stored in the database. The main advantage of BIM over other modeling methods is the availability of information about the building or structure, which can be used at different stages: architectural design, engineering calculations, construction, maintenance, reconstruction [2].

The opportunities of software systems that implement the technology of BIM, such as Autodesk Revit, Archicad, Allplan, AECOSim, are expanding every year. However, universal application that would solve all tasks throughout the life cycle of the building does not exist.

An important step in the design of the building is structural analysis. There are many powerful software products, that allow to carry out strength analysis based on the finite element method: LIRA, Ansys, SCAD, etc. Most of them were developed before the appearance of BIM technology, therefore the standard means of sharing information with these programs are either defective or absent. So there is the problem of interoperability between models created in BIM application and data of structural analysis software [4].

There are several variants of solution. For example, the ACIS technology can be used to transfer data from Autodesk Revit to Ansys. ACIS technology exports 3D geometric model to text file format \*.SAT (Save As Text). Ansys supports this format, but a number of problems arise when Ansys reads data. First, geometrical information is not sufficient for the strength analysis. Secondly, when Ansys creates a finite element system for structural analysis the finite element scheme has many inconsistencies in the vertices of elements. We have quite a lot of work to do manually to adjust the exported model.

IFC (Industry Foundation Classes) is another format for transferring data between programs. It is one of the most useful technologies, which can be applied to Building information modeling [3]. The IFC data have text format and contain information not only about geometrical, but also about physical and other properties of the model. Despite the fact that IFC holds almost all the information about the model, there is still a need to correct some information manually. Besides, many applications, such as Ansys, do not support this format at all.

## 2. Transferring data realization

One of the ways to automate the design process is creating a plug-in for data converting. The purpose of this plug-in is to convert the IFC file to a format supported by strength analysis software. Although IFC is an open format, the data is stored in encrypted form and the decryption process is quite time-consuming. In addition, not all the information about the model, but only certain parameters are required for structural analysis.

In this paper the problem to develop a new data format was stated and solved. We introduce method to automatically generate a file that contains all necessary information for the further strength analysis. Autodesk Revit is software which supports BIM and generates and stores information about building structures. Revit has built-in features of the API for developing plug-ins in the C# language in Microsoft Visual Studio. The building information model from Revit can be exported to any structural analysis program which provides the ability to load source data from a text file. This article describes the conjugation of Autodesk Revit with original IRA 2.0 application (developed by the authors) on the basis of the finite element method (FEM) [1].

The scheme (Fig. 1) shows the process of transferring data from Revit to the structural analysis program IRA 2.0. The plug-in (also developed by the authors) gets all Revit model parameters (geometrical, physical and logical) from the database and writes them into a text file. A special program-converter uses this file to convert all required parameters into a data format that is supported by structural analysis program (in this article, IRA 2.0).

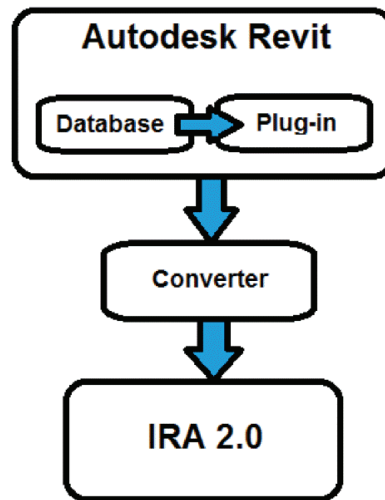


Fig. 1. The scheme of transferring data from Revit to structural analysis program IRA 2.0

We demonstrate the whole process on the following example. We consider a multi-frame consisting of three columns and two beams with I-shaped profile. The information model of this frame, which is created in Revit, is shown on the Figure 2.

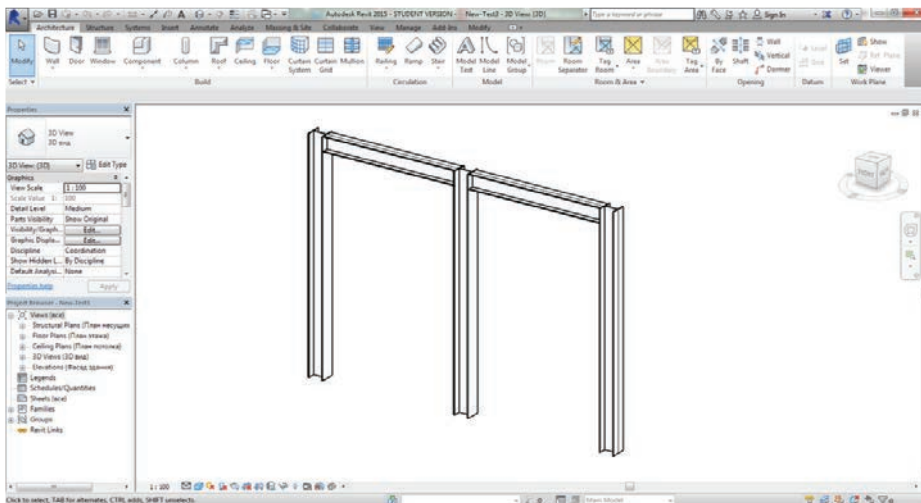


Fig. 2. The model of multi-frame created in Revit

If the user has pressed specially added to Revit toolbar button, the plug-in is started. The button is located on the tab "Add-Ins" ribbon in the "External" Revit panel. The work of the plug-ins implemented in the interactive mode where the user can select one or more of the structural elements or the building (Fig. 3)

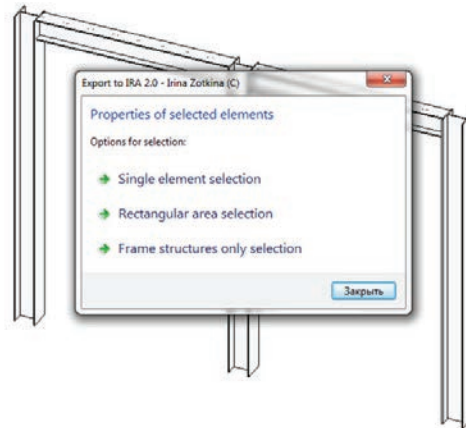


Fig. 3. Revit menu for selecting the elements

The result of the plug-in operation is a text file that contains parameters for all selected structural elements of the building (Fig. 4 a), including those parameters that are necessary for subsequent structural analysis (Fig. 4 b)

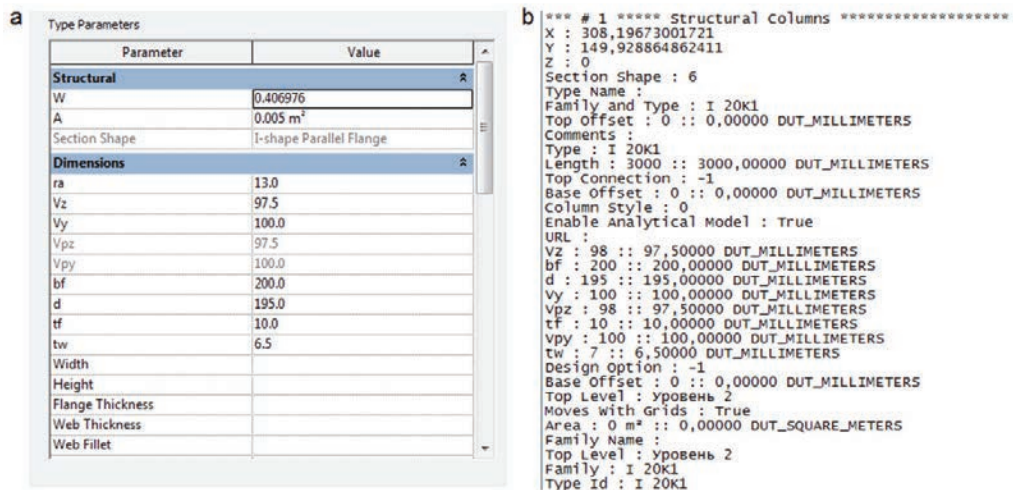


Fig. 4. Parameters of elements (a) in Revit; (b) in text file

The next step of the interoperability process between Revit and IRA 2.0 is the start of program-converter (also developed by the authors). The converter was developed in C++ as console application. The purpose of the converter is to read a file created by the Revit plug-in, and then to select and to transform required parameters into compatible with structural analysis software IRA 2.0 format. Then the converter automatically runs IRA 2.0 which generates FEM model of selected Revit constructions. Within IRA 2.0 program the user can attach different loads, adjust the geometrical and physical design parameters and carry out strength analysis by FEM (Fig. 5).

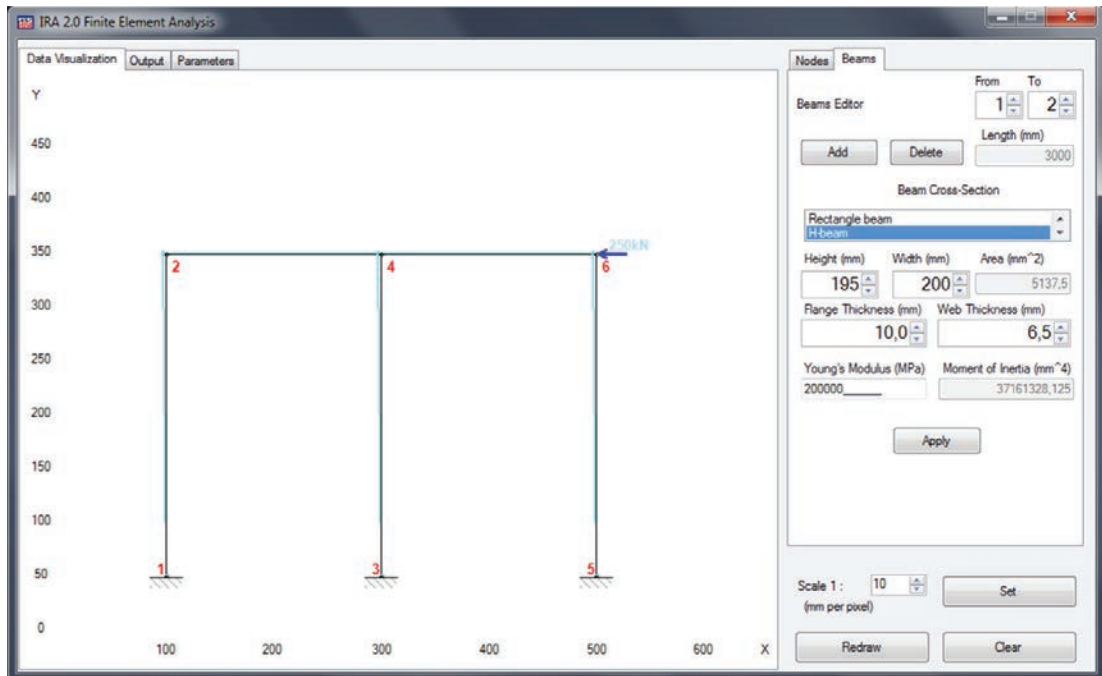


Fig. 5. FEM model in IRA 2.0.

## Conclusion

The described method and developed software (Revit plug-in, converter, IRA 2.0) allow to provide interoperability between BIM program (Autodesk Revit) and structural analysis software (IRA 2.0).

## References

- [1] Zotkin S.P., Blokhina N.S., Zotkina I.A. (2015) About development and verification of software for finite element analysis of beam systems, *Procedia Engineering* 111 (2015) pp. 902 – 906.
- [2] Ignatova E.V., Ignatov V.P. (2011) The effective use of building information model, *Vestnik MGSU* 1/2011, pp. 321-324.
- [3] Ignatova E.V., Kirschke H., Tauscher A., Smarsly K. (2015) Parametric geometric modeling in construction planning using industry foundation classes, *Digital proceedings of the 20th international conference on the applications of computer science and mathematics in architecture and civil engineering* (2015) pp. 68 –75.
- [4] Kukushkin I.S., Aloyan R.M., Marabaev N.L. (2015) Analysis of the use of bilateral integration of data between CAD and CAE in a medium BIM, *Informacionnaya sreda VUZa, Ivanovo State Polytechnical University*, pp. 132-143.