DATA TRANSFER BETWEEN STRUCTURAL ANALYSIS AND ASSESSMENT SOFTWARE

Alena Horská, *

Department of Concrete and Masonry Structures, Faculty of Civil Engineering,
Czech Technical University in Prague, Thakurova 7/2077, 166 29 Prague 6, Czech Republic.
alena.horska@fsv.cvut.cz

ABSTRAKT

V prostředí MS Excel byl vytvořen nástroj Robot-to-FIN pro přenos dat mezi Robot Structural Analysis a FIN EC. Vytvořený produkt najde své uplatnění při návrhu železobetonových konstrukcí a umožňuje inženýru rychle a efetivně exportovat výsledky statické analýzy a vkládat je do softwaru pro statické posouzení. Automatizace tohoto procesu je v případě velkých a složitých konstrukcí nutností a kromě značné úspory na výpočetním čase také snižuje riziko vnesení chyby. Tento článek prezentuje důvody, proč je vhodné oddělit proces analýzy vnitřních sil od procesu posuzování, demonstruje, jak složitost výpočetního modelu ovlivňuje objem přenášených dat, a popisuje nástroj vytvořený pro automatizaci přenosu dat.

KLÍČOVÁ SLOVA

Robot Structural Analysis • FIN EC • Přenos dat

ABSTRACT

Robot-to-FIN is a tool for data transfer between Robot Structural Analysis and FIN EC that was prepared in Excel spreadsheet environment. The product finds its use in the process of concrete structures design and enables the user to export results quickly and efficiently from structural analysis software and import them into FIN EC software to be assessed. Automation of such process is a necessity when analysing large and complex concrete structures, and apart from significant reduction of working time it also brings human error mitigation. This paper outlines reasons why it is desirable to separate the structural analysis from the assessment, shows how complexity of the structure effects the demands on data transfer, and finally presents the product that was prepared for automation of the transfer process.

KEYWORDS

Robot Structural Analysis • FIN EC • Data transfer

1. INTRODUCTION

One of the most frequent processes of design in structural engineering practise includes following steps: Preliminary design of geometry, materials and dimensions; Structural modelling in Finite Element Method (FEM) software; Obtaining internal forces, stresses and displacements; Determining structural resistance and serviceability limits of all structural parts; Assessment of Ultimate Limit State (ULS) and Serviceability Limit State (SLS); Optimisation of the design.

---

* Supervisor: prof. Ing. Alena Kohoutková, CSc., FENg.

Many FEM software products perform not only structural analysis, but also have integrated a tool for direct ULS and SLS assessment, and sometimes even generate a report, condensing several steps of the procedure into one. While this functionality makes the whole design process easy and comfortable, there are still reasons why the designer might tend to choose performing the assessment independently from the analysis, using different software products: Some of the reasons follow:

- An experienced designer does not blindly rely on the results obtained from the analysis (internal forces, stresses, displacements), but critically evaluates their validity, uncovering potential errors in the modelling. Such control over the results is possible only if there is a mid-step between analysis and assessment. If both analysis and assessment are performed in one step, errors in model remain hidden, as it is less obvious to spot them without seeing the internal forces and displacements.

- The ULS and SLS functionality may contain flaws when solving complex structures with complicated geometry. It was observed that some software products are suitable only for assessment of simple building structures that consist of horizontal rectangular slab elements and vertical rectangular wall elements. Assessment of structures with inclined or curved surfaces had unrarely severe errors.

- FEM results are sensitive to several factors, such as mesh type and size or level of geometry/load/support idealization. For that reason, some areas of the model are prone to show unreasonable values of internal forces. An experienced designer detects those areas, evaluates the validity of internal forces, and proposes possible adjustments of their values before performing ULS and SLS assessment.

If the structural designer decides to perform the analysis and assessment separately in two different software products for one of the reasons above, they will need to solve the issue of data transfer. The output of the structural analysis (internal forces, stresses, displacements) needs to be taken as an input for the assessment. This process can be easily done manually for simple and small structures with low amount of structural elements and load combinations. However, the results on more complex structures contain huge amount of data that cannot be transferred manually. In such cases an automation of the data transfer is required. Following section provides example structures demonstrating the data amount for structures of varying complexity.

2. DATA AMOUNT

Three example structures assessed for ULS are compared in this section to show how the amount of analysis results data changes

with changing complexity of the structure. The first example is a simply supported single span beam loaded with 1 load combination modelled in 2D, see Fig. 1. Such simple structure does not even require FEM modelling and can be analysed manually. There is no need for detailed assessment of the whole beam length, as it can be easily detected that only two positions are determining the ULS assessment - mid-span and beam edge. Two internal forces are relevant in this model - bending moment M and shear force V. The total amount of data to be obtained from the analysis and used for the assessment is 4 values (2 internal force values in 2 positions for 1 load combination; 2 x 2 x 1 = 4).
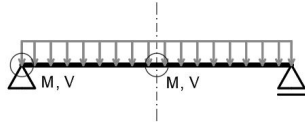
Figure 1: *Structural model no. 1.*

The second example is 2-storey frame structure loaded with 4 load combinations modelled in 2D, see Fig. 2. In this case also manual data transfer is chosen, which means that, again, not all positions of the structure are assessed, but only those obviously determining the ULS assessment. These positions are: edge and middle of each floor, top and bottom of each wall (in total 8 positions). Three internal forces are relevant in this model – axial force N, bending moment M and shear force V. The total amount of data to be obtained from the analysis and used for the assessment is 96 values (3 internal force values in 8 positions for 4 load combinations; 3 x 8 x 4 = 96).
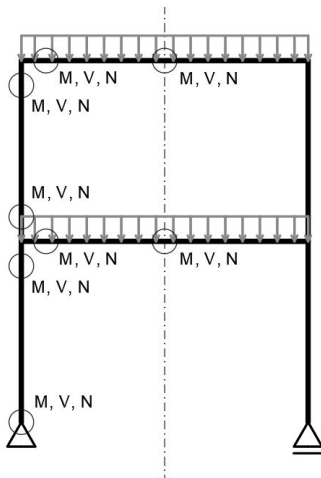

Figure 2: *Structural model no. 2.*

The third example is a segment of tunnel structure modelled in 3D, loaded with 19 load cases, creating the total of 59 load combinations, see Fig. 3. It is clear that this example structure is much more complex than the previous ones. Since it is not possible to visually spot all the positions determining the design, it is necessary to perform the assessment in all 6733 nodes of the FEM mesh. Eight internal forces on planes are relevant in this model – axial force N in two perpendicular directions, design bending moment M in two perpendicular directions at top and bottom surface of planes and shear force V in two perpendicular directions. The total amount of data to be obtained from the analysis and used for

the assessment is 3 177 976 values (8 internal force values in 6733 positions for 160 load combinations; 8 x 6733 x 59 = 3 177 976).
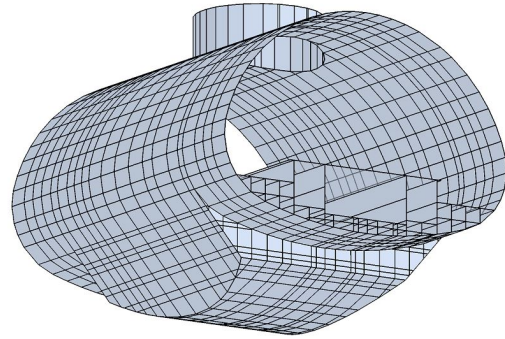

Figure 3: *Structural model no. 3.*

The third example structure shows that, in case of complex structures, the amount of internal force values is impossible to be transferred manually between the structural analysis software and the assessment software. Certain automation of the data transfer is therefore required.

3. DATA TRANSFER AUTOMATION

In order to automate the process of data transfer between the structural analysis software and assessment software, Robot-to-FIN tool was developed in MS Excel environment. Robot-to FIN was custom tailored specifically for data transfer between Robot Structural Analysis and FIN EC in the process of reinforced concrete structure design. A brief description of both software products is provided in following sections to show what type of data they work with.

3.1. Robot Structural Analysis

Robot Structural Analysis by Autodesk is a software used for analysis of structures using FE methods (Autodesk 2020). The software offers typical range of functionalities that most of structural FEM software product have. The goal of most users is to obtain internal forces and displacements on modelled elements. The internal forces, displacements and other results can be shown graphically as diagrams for 2D elements and maps for 3D elements. Robot Structural Analysis also provides the results in a tabular form as values for each node of FE mesh. The designer can use various filters on what elements and load combinations they wish to plot. The results can also be obtained either for each specific load case/combination individually, or as an envelope of combinations, that is frequently used functionality for structures with large number of load combinations. The user can also filter what results (forces, stresses, displacements, etc.) are shown in the result table.

Fig. 4 shows a typical table of results for one structural element. The values relate to envelope of several load combinations. For each internal force, the upper and lower extreme value is selected by the software, marking the upper extreme value with "»" symbol and lower extreme value with "«" symbol. The corresponding values of the remaining internal forces are assigned to the extreme value in the same row of the table. For example, the upper extreme of shear force in x-direction on panel 15 occurs in node no. 4156 for load combination no. 25 (the fifth row in the table). For that, the rest of the values in the fifth row show values of

Panel/Node/Case	NXX (kN/m)	NYY (kN/m)	QXX (kN/m)	QYY (kN/m)	MXX+ (W&A) (kNm/m)	MXX- (W&A) (kNm/m)	MYY+ (W&A) (kNm/m)	MYY- (W&A) (kNm/m)
15/ 3493/ 25	884.90>>	26.75	-175.01	-203.63	38.51	-30.55	29.17	-39.90
15/ 4042/ 25	-525.97<<	-825.63	210.92	-427.82	93.42	-185.65	200.34	-78.74
15/ 3434/ 25	140.01	390.23>>	-4.53	-155.81	0.0	-33.41	9.13	-17.63
15/ 4042/ 25	-525.97	-825.63<<	210.92	-427.82	93.42	-185.65	200.34	-78.74
15/ 4156/ 25	-347.00	-638.67	470.04>>	-257.87	0.0	-140.92	0.0	-98.48
15/ 3307/ 25	208.51	-583.08	-204.94<<	-77.69	0.0	-108.83	0.0	-336.32
15/ 3377/ 25	332.10	-389.68	-100.34	659.45>>	0.0	-71.03	0.0	-124.34
15/ 4149/ 25	60.50	-638.34	9.92	-667.63<<	124.10	0.0	210.25	0.0
15/ 3145/ 25	340.47	-219.50	148.17	-8.12	253.44>>	0.0	136.95	0.0
15/ 1583/ 25	133.92	-214.45	-26.81	-130.44	0.0<<	-52.73	0.0	-79.36
15/ 1584/ 26	1.14	-76.22	-0.92	8.33	4.07	0.0>>	7.87	0.0
15/ 3322/ 25	227.75	-598.37	-74.82	-238.24	0.0	-220.37<<	0.0	-592.90
15/ 4148/ 25	72.06	-632.50	-76.20	-646.55	190.79	0.0	250.75>>	0.0
15/ 1583/ 25	133.92	-214.45	-26.81	-130.44	0.0	-52.73	0.0<<	-79.36
15/ 1583/ 26	-1.64	-86.57	5.68	7.95	1.03	-1.46	3.51	0.0>>
15/ 3322/ 25	227.75	-598.37	-74.82	-238.24	0.0	-220.37	0.0	-592.90<<

Figure 4: *Table of results - Robot Structural Analysis.*

the other (non-extreme) internal forces in node no. 4156 for load combination no. 25.

The values of the result table can be copied out and inserted into MS Excel. However, the formatting is inconvenient and requires modification before next use of the data. Since large structural models require multiple repetition of the format modification, automation of that process is a necessity.

3.2. FIN EC

FIN EC by Fine is a software used for ULS and SLS assessment of various types of structures, including reinforced concrete (Fine 2020). The internal force values for assessment can be input into the software manually one by one, however, for large amount of values, it is more convenient to import them all at once. The software accepts the data specifically arranged in .txt format, see Fig. 5. Each row stands for one set of internal force values in one node of the structural model. The sequence in each row is following: Name of load case in double quotes – normal force – bending moment – shear force – considering 2nd order effects – code of load combination type – coefficient of load duration. The text order within a row and format of the .txt file must be strictly kept, otherwise FIN EC does not accept the imported file.

Figure 5: *FIN EC import file format.*

3.3. Robot-to-FIN

As the Fig. 4 and 5 show, the form of the data exported out of Robot Structural Analysis and data imported into FIN EC differs significantly. Robot-to-FIN was prepared to allow the designer to change the data format easily, using MS Excel. All steps that the user takes are described as follows:

1. Copy the required results table from Robot Structural Analysis and paste into the input field of Robot-to-FIN, see Fig. 6. Two input fields are prepared - for ULS and SLS values;

2. Click on "Fill output" button – the format of the exported data is modified and the values are arranged in text strings;

3. Click on "Copy out" button – all rows of text are copied out;

4. Paste into Notepad file – the .txt file is created ready for import to FIN EC.

Figure 6: *Robot-to-FIN input field.*

As described above, the whole data transfer is operated by copy-paste command and several buttons that run macros programmed in Visual Basic (VBA) (Vaswani 2018), see Fig. 7. The processes hidden under each button are described below.

- Fill output: Erases "«" and "»" symbols from the cells of extreme internal force value. Separates x- and y- direction of internal forces. Modifies the 'general' format of the cells to 'number'. Rearranges the table columns to obtain the

required sequence. Modifies the 'number' format of the cells to 'text'. Creates text strings fitted in one column from individual columns.

- Copy out: Stacks columns of ULS values and SLS values into one for both x- and y-direction without any blank rows, respecting the fact, that each structural element can have different number of nodes, and therefore, for each element the number of rows in the column differs. Copies out the data.

- Clear input: Clears the whole input field after finishing data transfer of one element to ensure that no values get mistakenly mixed with values of another element.

- Clear output: Clears the output field for the same reason as stated above.

- Clear all: Clears both input and output fields for the same reason as stated above.

Figure 7: *Robot-to-FIN command buttons.*

Apart from the basic principle and purpose described above, Robot-to-FIN offers additional features making the use more practical and convenient. As mentioned in Introduction, FEM analysis often gives unreasonable results, which is really only the effect of model simplification and meshing. For that the designer might sometimes choose to reduce some extreme values of some internal forces, for example peak bending moments on slabs above columns. This is possible in Robot-fo-FIN by defining limit values for each internal force, separately for ULS and SLS, see Fig. 8. When the limit values are defined, all values exceeding them are reduced to the limit before the .txt file is composed. However, it is strongly advised that setting the limits is done only after careful consideration of an experienced designer.

Figure 8: *Internal force peak reduction in Robot-to-FIN.*

For convenience of the designer, instructions for correct use are provided directly in the spreadsheet. Also, identification field is prepared for the user to fill in order to record the properties description of the subject part of the structure.

4. CONCLUSIONS

Robot-to-FIN tool for data transfer between Robot Structural Analysis and FIN EC was prepared in Excel spreadsheet environment. Robot-to-FIN finds its use in the process of concrete structures design and enables the user to export results quickly and efficiently from structural analysis software and import them into FIN EC software to be assessed.

References

Autodesk (2020), *User's guide to Robot Structural Analysis*.

Fine (2020), *FIN EC 2021 - Uživetelská příručka*.

Vaswani, K. (2018), *Udemy online course: Excel Automation: Lear VBA Macros From Scratch*.