A Structural Simulation System, Part 1: Interface and Operation

Duan Jin[1,a], Chen Xiao-ming[1], Li Yun-gui[1]

[1] China State Construction Technical Center, Beijing, 101300, PR China

[a] duanjin78@126.com

Key words: structural simulation, system integration, finite element analysis

Abstract: This paper presents an integrated simulation system for building structures. It is to satisfy the engineering requirements for high performance simulation and is a coalition of the traditional design software and general finite element analysis (FEA) software. Based on this system , the designed models, obtained by traditional design software, such as Pkpm, Yjk etc., could be automatically transformed to corresponding FE models and imported into commercial FEA software, such as Abaqus, Ansys etc., to finish the complex FE analysis.

Introduction

According to the more and more popularity of complex buildings, such as complicated shape building, high-rising building, long-span structure etc., the high performance simulation has become more and more import in the design and construction of building structures. The traditional design software, such as Pkpm, Yjk etc., could not satisfy the above requirements due to the deficiency of computation. On the other hand, the commercial FEA software, such as Abaqus, Ansys etc., although have powerful calculated performance, while they are not convenient for building structure analysis due to the fact that their preprocessors are not suitable for building modeling. Therefor from the civil engineering perspective, a simulation system simultaneously having the advantages of traditional design software and commercial FEA software is very useful even desired to solve the practical engineering problem.

In this paper, an integrated simulation system for building structures, or ISSS for short, is presented to satisfy the building engineering requirements for high performance simulation. It is a coalition of the traditional design software and general finite element analysis (FEA) software. Based on this system , the designed models, obtained by traditional design software, such as Pkpm, Yjk etc., could be automatically transformed to corresponding FE models[1-4] and imported into commercial FEA software, such as Abaqus[5], Ansys etc., to finish the complex FE analysis.

Interface of the simulation system

Figure 1 presents the main interface of integrated simulation system (ISSS) and Figures 2~5 presents some options of parameter setting of ISSS. In ISSS, the traditional structure-design software and the general finite element software are united to an integrated system, which includes the following capabilities and some others:

- Interface for structural design software: obtain the structural design data, including geometry and reinforcement, from the traditional structure-design software, such as PKPM, YJK, MIDAS etc., see Figure 2 for YJK;
- Interface for finite element model generation: transform the structural model into FE

calculation model automatically, including quadrilateral meshing, cross section setting, load calculation, restraint and connection management, etc., see Figure 3;

- Interface for commercial FEA software: automatically input the above FE calculation model into the commercial FE analysis software, such as ABAQUS, ANSYS etc., to perform the finite element analysis with some secondary development for concrete constitution model and finite element model, see Figure 4 for ABAQUS;

- Interface for result report: read out the computing results from commercial FEA software, and calculate the structural performance parameters, and evaluate the structural damage and security, and finally automatically generate the computing report, see Figure 5.
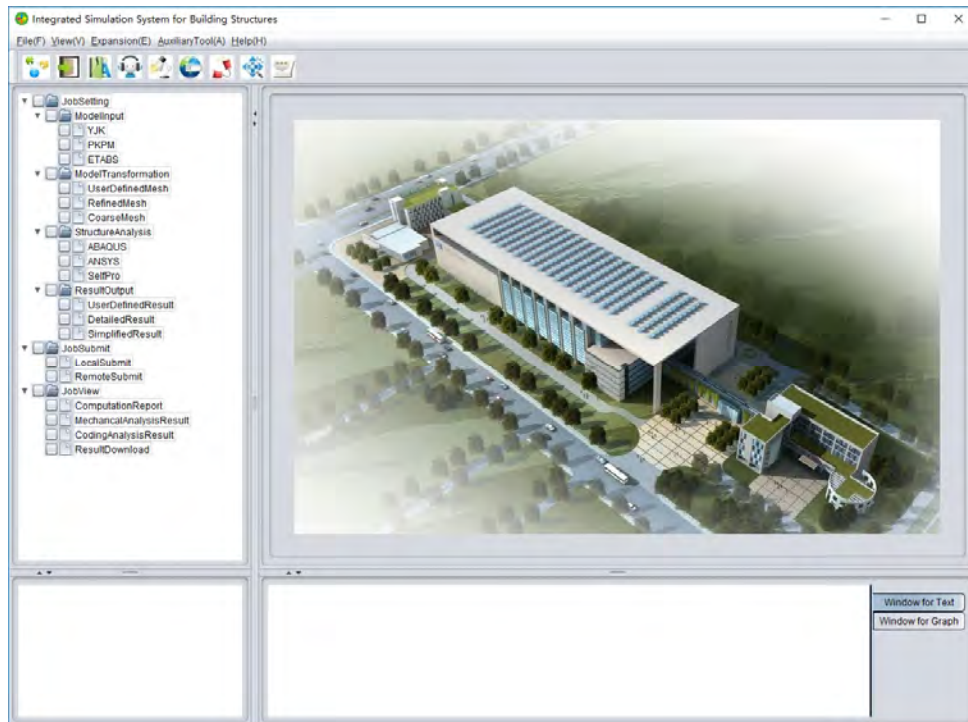
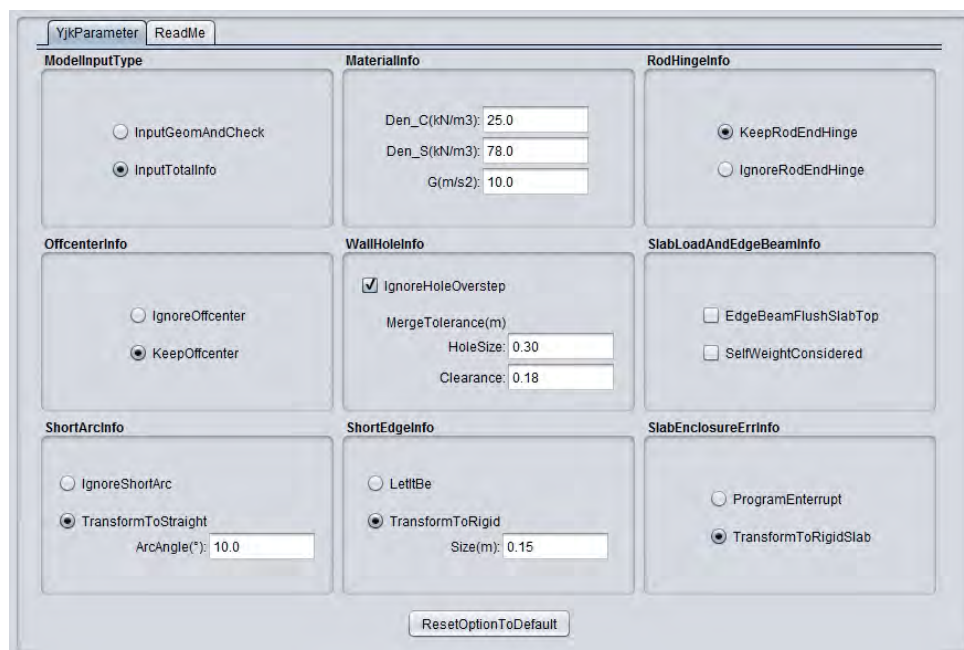Figure 1 Interface of the integrated simulation system
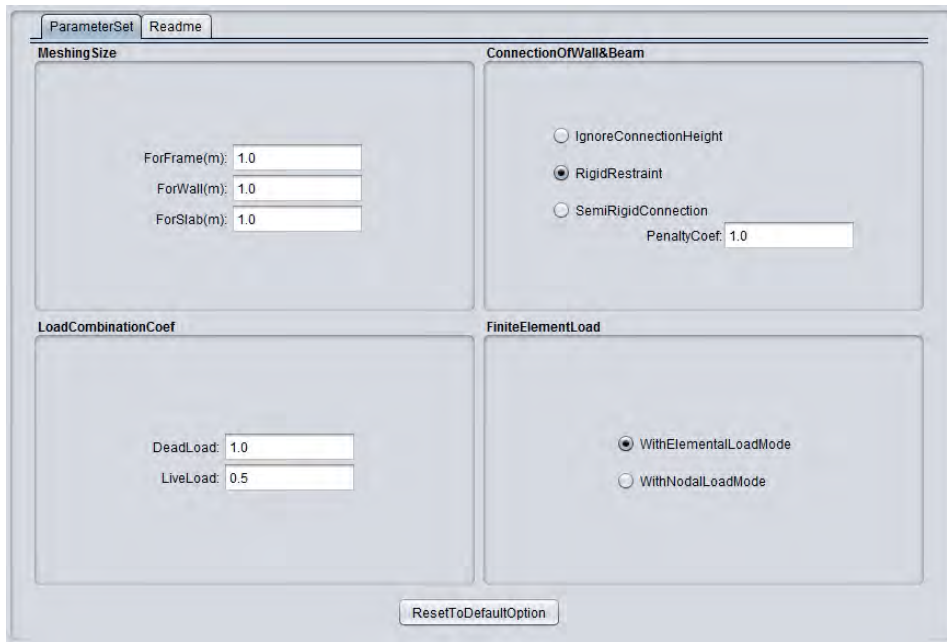
Figure 2 Parameter Setting of YJK data input

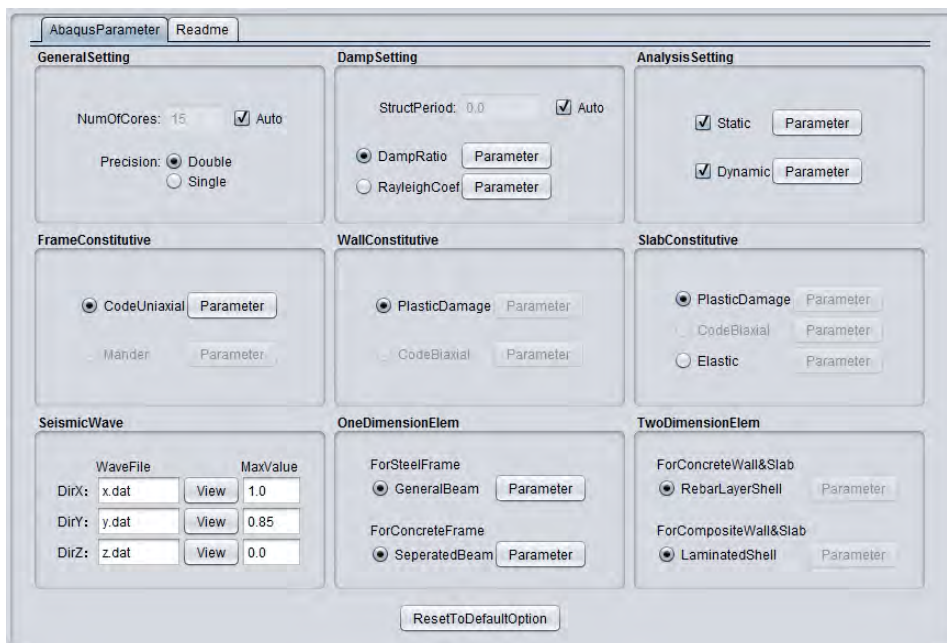Figure 3 Parameter Setting of FE model generation

Figure 4 Parameter setting of ABAQUS calling

Operation steps of the simulation system

The core idea of integrated simulation system is to realize the automatic operation for large structure analysis. It mainly includes the following operation steps:

Step 1 (modeling and general design): finish the structural modeling and general design with the traditional design software, such as PKPM, YJK etc. This step is just as the standard procedure of structural design and is not associated directly with the integrated system other than providing the structural design data;

Step 2 (parameter setting for simulation analysis): this step includes the following four

parts. Firstly, use the interface for structural design software to read in the designed model, including the reinforcement information, see Figure 2. Secondly, use the Interface for finite element model generation to generate the FE calculation model automatically, including quadrilateral meshing, cross section setting, load calculation, restraint and connection management, see Figure 3. Thirdly, use the interface for commercial FEA software to call the computing kernel of ABAQUS, ANSYS, or others, see Figure 4 for ABAUQS. Fourthly, use the interface for result report to read out the computing results from commercial FEA software, and calculate the structural performance parameters, and evaluate the structural damage and security, and finally automatically generate the computing report, see Figure 5;

✚ Step 3 (simulation job submit): submit the simulation job with the above parameter setting. The integrated simulation system would successively read in the structural designed model, generate finite element mesh, conform the structural load, set the structural restraint, call the computing kernel of commercial FEA software, read out the computing result and finally generate the result report automatically.
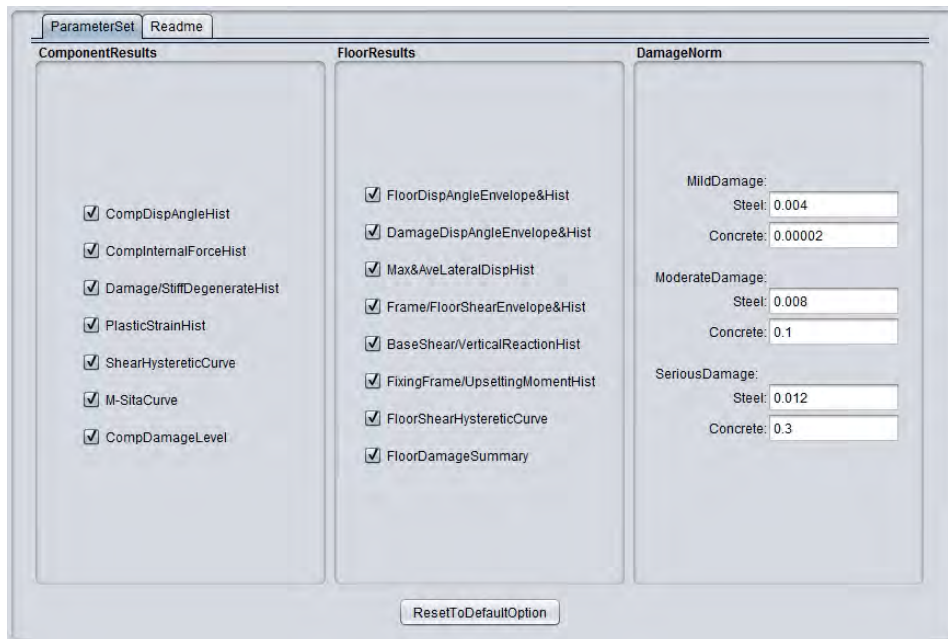

Figure 5 Parameter setting of user defined result output

Summary

The integrated simulation system is developed for the high performance computation of building structures, including security assessment and optimization design. The traditional structure design software and the commercial FEA software are integrated into an entirety with a mass of secondary development of software. The most striking feature of the integrated system is realizing the automation of composite structure simulation, including automation design model input, automation FE model generation, automation FE computation and automation result report.

Reference

[1] Li Y.G., Duan J. & Chen X.M. Simulation Analysis and Engineering Examples for Building Structures [M]. Beijing: China Building Industry Press, 2015. (in Chinese)

[2] Duan J., Chen X.M. & Li Y.G. A Quadrilateral Meshing Method for Shear-Wall Structures [J].

Applied Mechanics and Materials. 2014, 638-640: 9-14.

[3] Duan J., Chen X.M., Qi H. & Li Y.G. Boundary-Constraint Meshing Based on Paving Method [J]. Applied Mechanics and Materials. 2014, 627: 262-267.

[4] Duan J., Chen X.M., Qi H. & Li Y.G. An Automatic FE Model Generation System Used for ISSS [C]. Proceedings of Civil Engineering and Urban Planning III. Landon, UK: Taylor & Francis Group. 2014: 29-32.

[5] Abaqus 6.13 Theory Manual. Providence [R]. RI, USA: Dassault Systèmes Simulia Corp, 2013.