

Integrative structural design

Fabian Gerold

Bauhaus University Weimar, Germany

Abstract

In the conceptual design stage, numerical simulations are often not applied, although they could give valuable feedback about important aspects of structural performance, like robustness, redundancy, ductility. Current structural design software is not optimized for conceptual design; therefore, structural designers are not encouraged to use numerical simulations throughout the structural design process.

In this project, a new approach to the use of numerical simulation in conceptual structural design is made, using the possibilities of today's hardware and open source software frameworks. The resulting structural design tool for conceptual, interactive modelling, simulation and visualization is presented under the term "Integrative Structural Design System" (ISDS).

Keywords: structural simulation, visualization, user interaction, re-analysis, instant feedback

1 Interactive modelling

When numerical simulation is used in the conceptual design stage, it is essential to be able to easily define and change system parameters. These could be geometry, topology, material and cross sections of structural members. Moreover, it is essential too, to get instant visual and other feedback from the structural behaviour after changes have been made. Traditionally, the engineer has to handle different applications for input, analysis and post-processing, but it is difficult to track consequences of design decisions and changes through different applications. The engineer often finds himself in an imposed sequential workflow, although conceptual design is a highly iterative process.

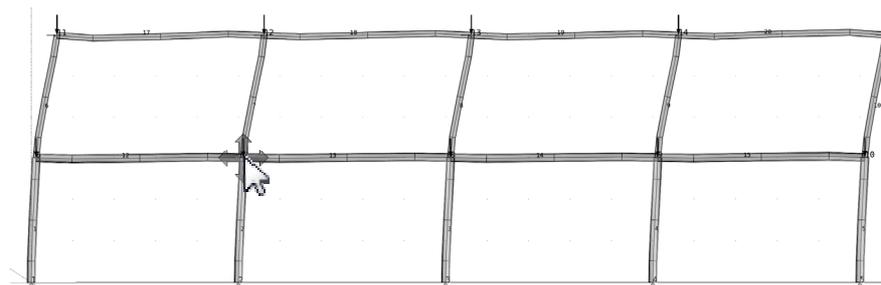


Figure 1. Interactive model of a frame system in ISDS

A different approach would be more intuitive: define and change the model in one application (figure 1), run the analysis in the background, and update the model visualization (structural response) after the analysis (or re-analysis).

In the example frame system (figure 1), that is further described in the next section, the following system parameters can be changed interactively:

- Node coordinates
- Nodal constraints
- Nodal loads, element loads
- Material and cross section properties
- Topology (members between nodes)
- Analysis parameters

Interaction with the model is most intuitive, when it takes place directly in a 3D viewer, for example dragging nodes with a pointer device, e.g. mouse (figure 2).



Figure 2. Dragging a node in the frame system (re-analysis time: 70 ms on a standard PC)

Using a client-server approach for the numerical (re-) analysis, the system results can be updated continuously during the dragging command (figure 2). This is further described in section 3.

A change in the topology also triggers an automated re-analysis, showing the structural response instantly (figure 3).

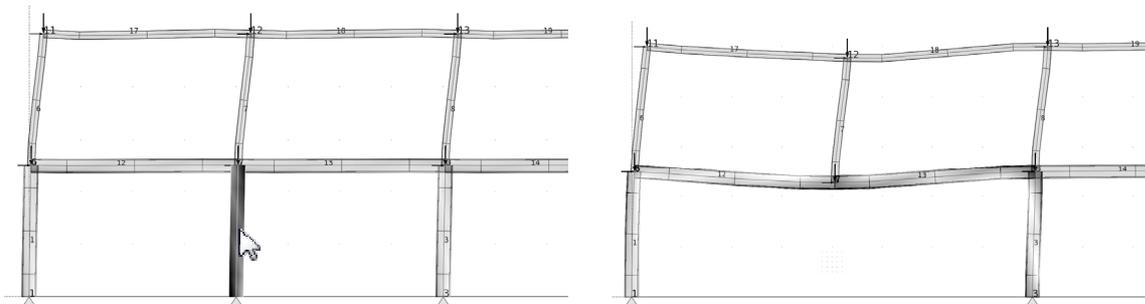


Figure 3. Selecting and removing a column in the frame system (re-analysis time: 80 ms)

For other parameters like nodal constraints, editing in the 3D viewer is not suitable. In this case, a table based input would be preferable.

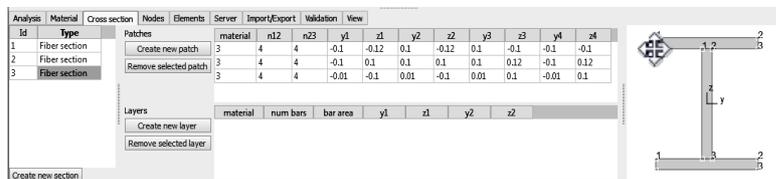


Figure 4. Cross section editing in ISDS

Cross sections could be edited with a CAD-like 2D interactive viewer (figure 4).

In ISDS, these functionalities are implemented on an experimental basis, using OpenSceneGraph (Martz and Kuehne, 2007) as 3D viewer and Qt (Blanchette and Summerfield, 2008) as graphical user interface framework. Further details about the software architecture are discussed in section 5.

2 Example frame system

A simple frame system was chosen to demonstrate possibilities of *Integrative Structural Design*. The structural system consists of beams and columns (figure 5), using nonlinear material models (figure 6). The span of the fields is 5.0 m, the height between floors is 3.0 m.

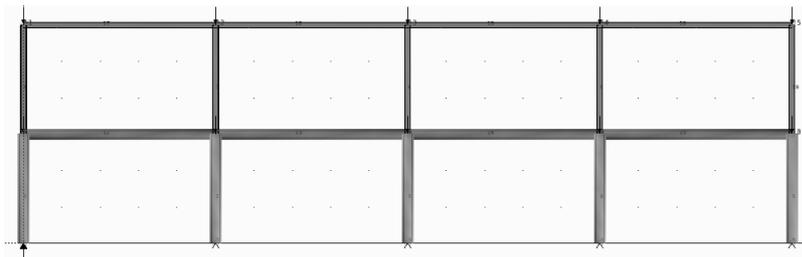


Figure 5. Example frame system

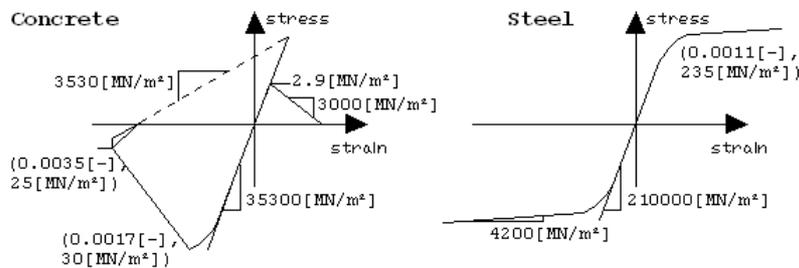


Figure 6. Material models for concrete and steel

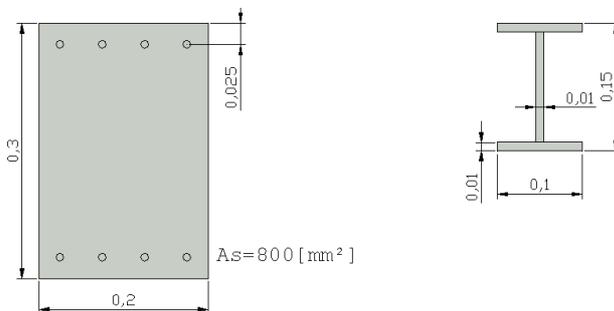


Figure 7. Cross sections for the first and second floor

The example frame application can be downloaded under www.isdserver.com

3 Instant numerical re-analysis

In ISDS, the Finite Element Method (FEM) is used for numerical analysis of the structural system. A structural response that is as close to reality as possible is essential. Therefore, nonlinear material models are applied (figure 6), using the advanced open source FE framework OpenSees (McKenna, 1997, Mazzoni et al., 2006).

Also very important is a fast analysis and re-analysis after changes have been made in the model (figure 2, 3). Therefore, it is necessary to perform a re-analysis with a system that is kept in memory, rather than the traditional input file approach. An input file requires to build up and prepare the complete system every time a change happens, even if that change only affects one number in the stiffness matrix. Ways to achieve efficient re-analysis are discussed in section 4. As pointed out before, the (re-) analysis starts automatically in the background, not disturbing further editing. Of course, automated re-analysis in ISDS can also be set to manual.

3.1 Synchronizing re-analyses

After an analysis, the graphical representations of the results are updated as soon as the analysis is done. If the model has been edited in the mean time, there are different possibilities: wait for the previous analysis, interrupt the previous analysis, or start another analysis and run both (or more) in parallel. Then update result visualization in the same order the analyses have been started. At the moment, only the first (and simplest) method is implemented in ISDS. Tests have shown, that this works well with systems up to 3000 degrees of freedom. Beyond that, parallelization is suitable, or one of the other mentioned approaches.

3.2 Visualization of dynamic analysis results

A real-time visualization for example of an earthquake time history, including displacements, stress/strain visualization, gives a good impression of the overall structural behaviour of a system. ISDS provides the possibility to asynchronously analyze and buffer the results using a client-server architecture that is described in the next section.

4 FE analysis in a client-server approach

As pointed out before, the traditional input file based FE analysis is not optimal for fast re-analysis, which is crucial for intuitive modelling. The input file could be avoided by adding the FE framework as linked library. But then the application becomes very complex and potentially unstable. Therefore, a client-server approach has been developed and implemented in ISDS (figure 8).

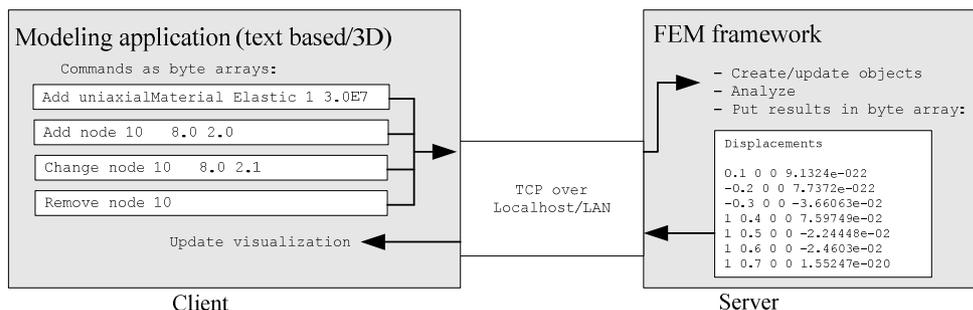


Figure 8. Client-server architecture of ISDS

As changes happen in the model, a “change” command can be easily sent to the server. The server applies the change to the existing FE analysis model, does a re-analysis and sends the results back to the client. So the analysis runs asynchronously to the main event loop of the client application.

A client application can be implemented with any language or on any platform that supports TCP. Even a web-based application could use an FE server over the internet.

4.1 Communication protocol

The reliable network protocol TCP was chosen for the communication between server and client. Data packages are used to send data and commands (figure 9).

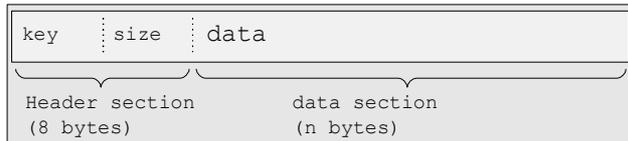


Figure 9. TCP data package

4.2 Validation of an FEM Server

A client application that connects to an FE server, should perform a validation before using the results that the server provides. Therefore, test systems with known analytical results are implemented in ISDS, and a plot allows a comparison between the results provided by the server, and the analytical results (figure 10).

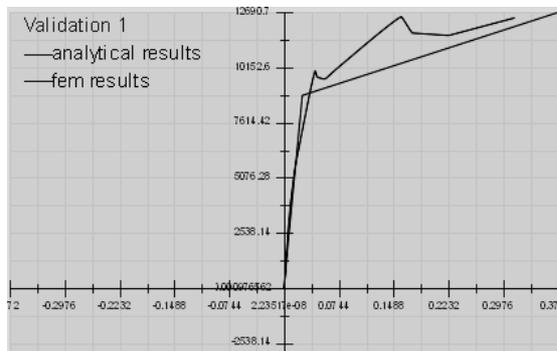


Figure 10. Plot of validation results in ISDS (pushover-analysis)

5 Software architecture

For research purposes like the above described frame application, a package of classes has been developed, to provide basic functionality like command management and user interaction. Several other packages are necessary to complete the application, shown in figure 11.

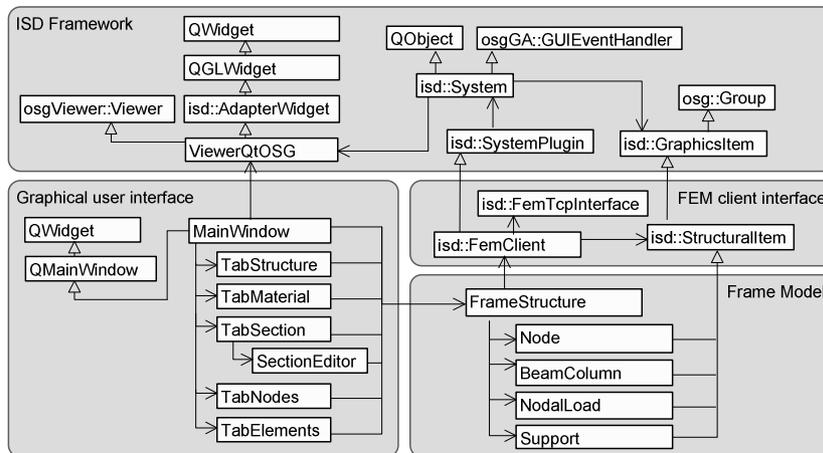


Figure 11. UML class diagram of ISDS and the frame application

6 Conclusions

The demonstrated interactive frame model, combined with instant re-analysis and visualization gives an intuitive experience of the structural behaviour of beam/column systems. So it can be useful in the early design stage, where the influence of system parameters and design decisions need to be evaluated quickly.

Also as eLearning application, ISDS could be helpful for understanding structural behaviour. Other eLearning simulation applications could be developed on this basis.

The client-server analysis approach has proven to be very efficient and flexible, providing platform independence, and reducing complexity for client applications.

References

- BLANCHETTE, J., SUMMERFIELD, M. (2008), *C++ GUI Programming with Qt 4*, Prentice Hall PTR
- MAZZONI, S., McKENNA, F., SCOTT, M. H., FENVES, G. L., et al. (2006), *OpenSees Command Language Manual*
- MARTZ, P., KUEHNE, B. (2007), *OpenSceneGraph Reference Manual v2.2*, Skew Matrix Software and Blue Newt Software
- McKENNA, F. (1997), *Object-Oriented Finite Element Programming: Frameworks for Analysis, Algorithms and Parallel Computing*, Dissertation
- OpenSees, Open System for Earthquake Engineering Simulation, <http://opensees.berkeley.edu/>
- OpenSceneGraph, <http://www.openscenegraph.org>
- Qt GUI Toolkit, <http://www.qtsoftware.com>