RFEM-Plugin for SIMULTAN (Work in Progress)

Sep 01, 2022

CONTENTS

This user guide should help navigate the RFEM-Plugin which enables structural analysis with the SIMULTAN datamodel based on the finite element software RFEM 6 by Dlubal.

- Inhalt
  - *Introduction*
  - *Getting started with the RFEM Plugin*
  - *Setting up a problem*
  - *Running a simulation*
  - *Results of the simulation*
  - *SIMULTAN Datastructure to incorporate the RFEM Data model*
  - *Copyright and license agreements*
  - *References*

CHAPTER

ONE

INTRODUCTION

1.1 What is the RFEM-Plugin?

The RFEM-plugin (in the following referred to as "the plugin") enables structural analysis with the finite element software RFEM 6 by Dlubal for the SIMULTAN-data model. This is achieved by exporting all necessary data for rudimental structural analysis into an XML-file which can be imported into RFEM 6.

1.2 Functionality covered by the RFEM-Plugin

In the current state of development the plugin is able to export:

- Geometry
- Material data
- Loads
  - Nodal
  - Line
  - Surface
- Supports with certain nonlinear behaviour
  - Nodal
  - Line
  - Surface
- Structural components and their hinges with certain nonlinear behaviour
  - Beams
  - Columns
  - Surfaces

What we are not able to do right now:

- Import results back into the SIMULTAN-data model (since in its current state there is now mesh visualization implemented in SIMULTAN-data model)
- Track changes made in the RFEM-data model but not in the SIMULTAN-data model

1.3 Project

The development of the RFEM-Plugin is the result of the FFG-research project "Ganzheitliche Gebaeudesimulation" supervised by Christoph Bauer. Additional information about the project and project outcomes can be found under these links:

- Ganzheitliche Gebaeudesimulation Press
- RFEM Plugin Publication
- IDA-ICE Plugin Publication
- Additional works

Information and publications about SIMULTAN:

- SIMULTAN as a Big-Open-Real-BIM Data Model - Proof of Concept for the Design Phase [PLW+19]
- Digital Twin applications using the SIMULTAN data model and Python [BuhlerSB22]
- SIMULTAN as a Big-Open-Real-BIM Data Model - Proof of Concept for the Design Phase [PLW+19]
- SIMULTAN - Simultane Planungsumgebung fuer Gebaeudecluster in resilienten, ressourcen- und hoechst energieeffizienten Stadtteilen [Betal]

1.4 Authors

Project supervision of the development of the RFEM-Plugin was done by Andreas Sarkany and Bernhard Steiner. Core development of the software was conducted by Zsombor Jarosi. Additional Feedback and testing was provided by Thomas Rabl and Thomas Bednar.

1.5 Getting help

You can get in touch with the authors via the Issue-Tracker of the repository this user guide is published on.

CHAPTER

TWO

GETTING STARTED WITH THE RFEM PLUGIN

2.1 Installing the SIMULTAN-Editor

To use the plugin you need to install the latest version of the SIMULTAN Editor, which can be found here. The SIMULTAN Editor enables you to interact with the SIMULTAN data model through a user interface. A comprehensive user-guide how to install and interact with the SIMULTAN Editor can be found under this link.

2.2 Installing the RFEM Plugin

After installing the SIMULTAN Editor you can install the RFEM Plugin via the Plugin Manager Fig. 2.1. Either download the plugin from the SIMULTAN Server (not possible yet) or install from your local disk with the installation file.

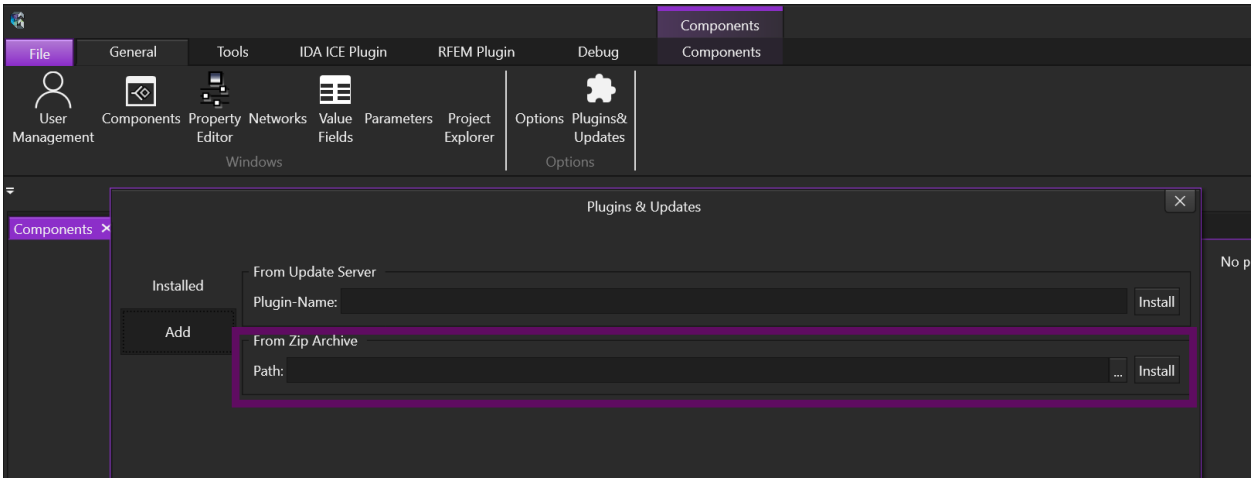

Fig. 2.1: Plugin Manager in he SIMULTAN Editor.

When the installation was successful a new tab will be added to your taskbar named RFEM Plugin Fig. 2.2.

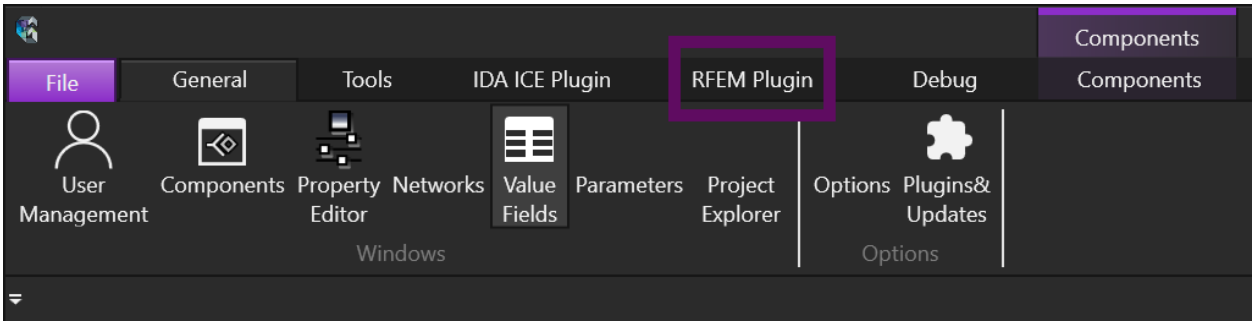

Fig. 2.2: New tab added to the taskbar of the SIMULTAN Editor after installing the RFEM Plugin.

2.3 Run the example

To check if everything was installed properly please run one of the examples provided where you downloaded the installation file of the plugin. Open the example in your SIMULTAN Editor (username and password is "admin"), and follow the instructions in the chapter *Running a simulation*. If this works fine and you are getting plausible results the installation was successful.

CHAPTER

THREE

SETTING UP A PROBLEM

A short overview of the representation of the data in the data model will be given in the beginning. After that the tutorial focuses on modelling with the implemented user interface of the plugin.

3.1 Structural analysis component

In Fig. 3.1 a simplified depiction of the structural analysis component is given. This component is used in the SIMULTAN datamodel to store the information needed to export data with the RFEM Plugin to enable structural analysis simulations in RFEM 6.

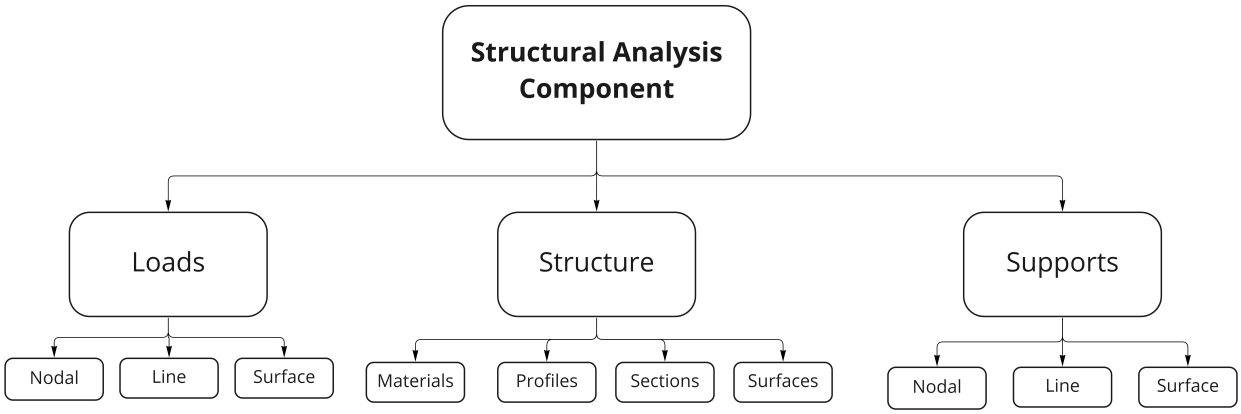

Fig. 3.1: Simplified depiction of the representation of the structural analysis component.

A more in depth look into the SIMULTAN representation of the structural analysis data is given in the chapter *SIMULTAN Datastructure to incorporate the RFEM Data model* .

Warning: Although this data structure can be implemented manually, it is strongly recommended to use the user interface that comes with the plugin for modelling. Manuel modelling is always prone to errors and could result in very time consuming efforts when trying to fix those.

3.2 Geometrical modelling

The geometry is created in the Geometry Editor of the SIMULTAN Editor. Please consult the SIMULTAN Editor User Guide for further information on how to use this Geometry Editor.

3.2.1 Modelling guidelines

In Fig. 3.2 it can be seen that the plugin uses a reference geometry (white) as well as a geometry for the structural analysis (red). The reference geometry, representing the outer building envelope, can for example be an architectural model or the model used for the building physics simulations.

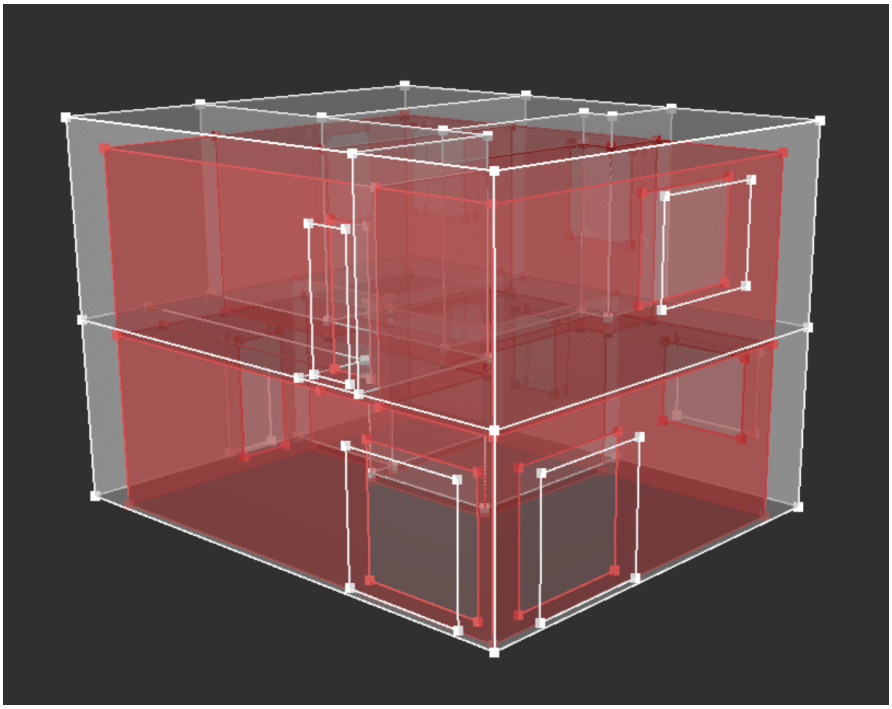

Fig. 3.2: Example of the geometry of a *Tiny House* modelled with the Geometry Editor of the SIMULTAN Editor.

Tip: The structural model (red) can be easily created by copying the surfaces of the reference geometry to the middle axis of the structural building components.

For the simulation the structural geometry is exported to RFEM, therefore all of the information exclusively used for structural analysis will be linked to this geometry.

3.3 User Interface

The RFEM-Plugin comes with a user interface Fig. 3.3 to help with the creation of the needed data structure for the plugin. The user interface itself is closely built around the graphical interfaces of RFEM 6. With this we hope to provide a familiar experience to all the RFEM users when using this plugin.

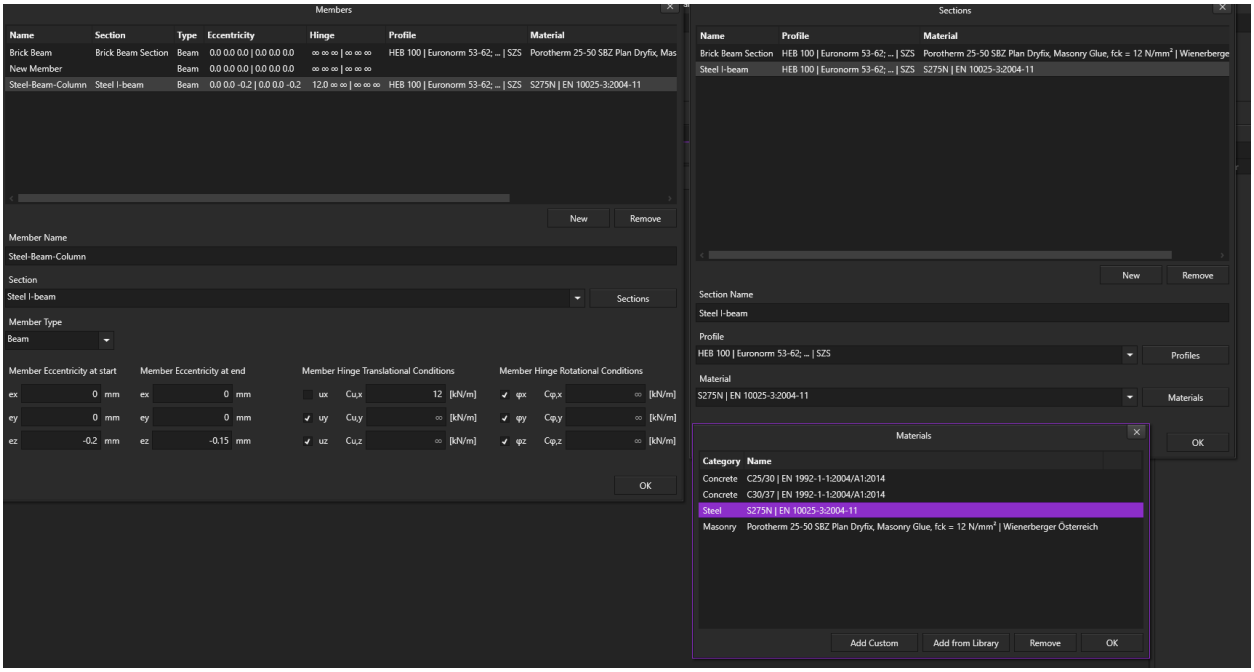

Fig. 3.3: Example of different layers of the user interface to create the data structure for the plugin when adding a new member.

3.3.1 Creating a new member

To create a new member simply click on `Member` in plugin's tab Fig. 3.4. The in Fig. 3.3 will open.

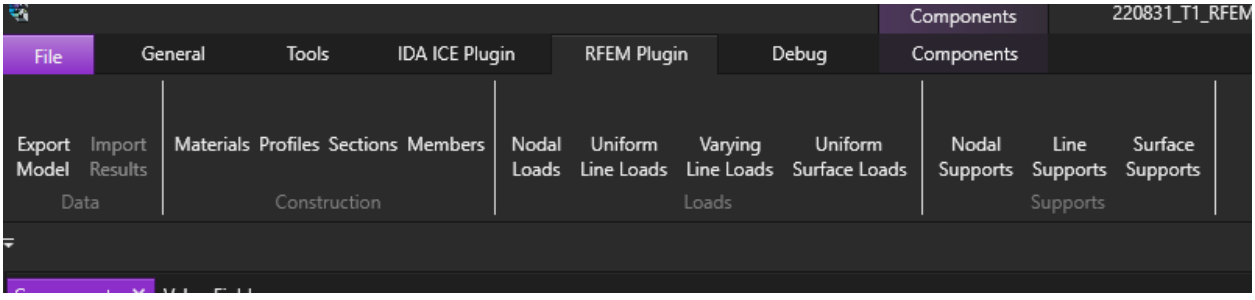

Fig. 3.4: Example of the plugin's tab where all the functions for the user-interface can be accessed.

Now follow these steps to create a new member:

1. Click on `New` to add a new member.
2. Select the newly created member, and change the name if needed.
3. Create a new section, by clicking on `Section` or choose an already existing section from the dropdown menu.

4. Choose the `Member Type` with the dropdown menu.
5. Adjust `Eccentricities` and `Hinges` if needed.

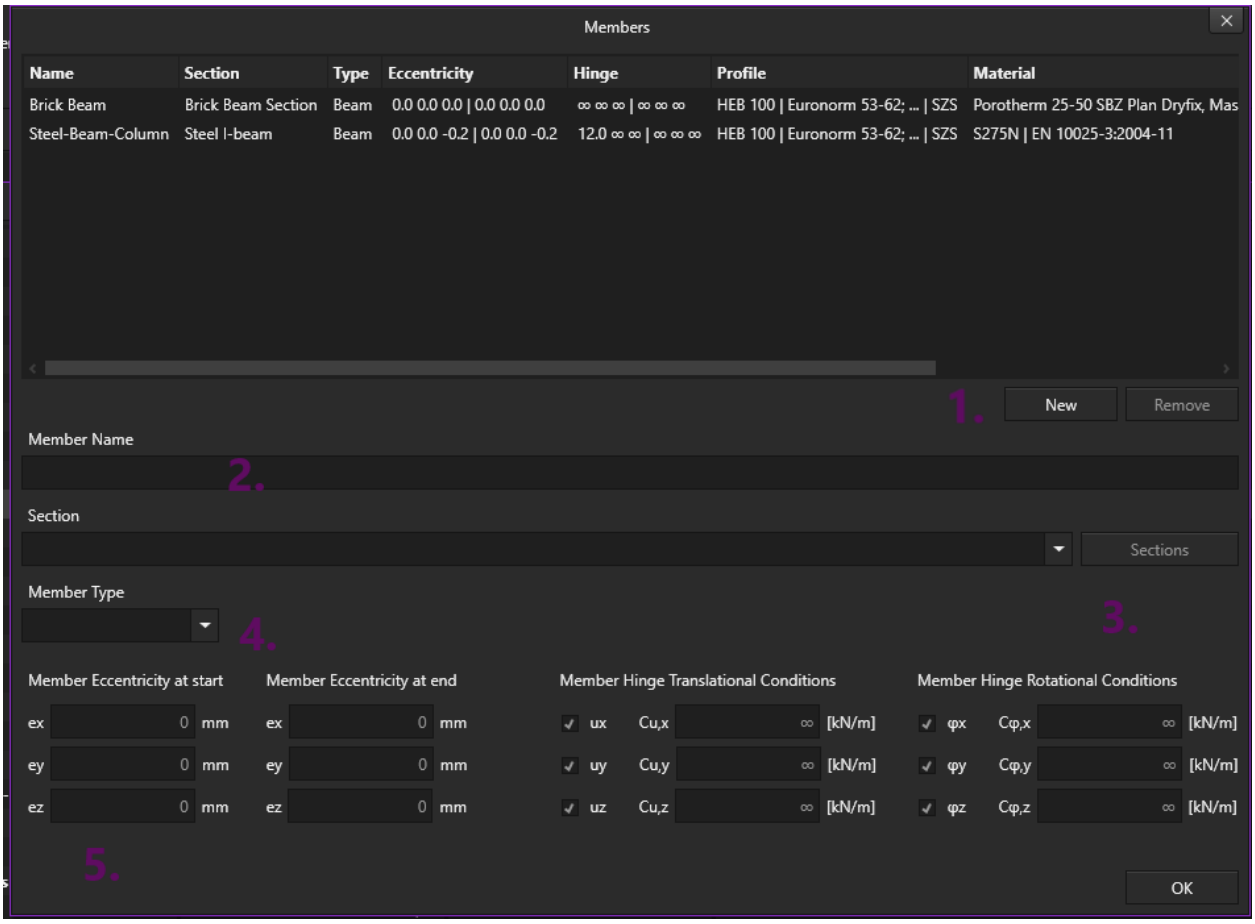

Fig. 3.5: Steps to create a new member in the data-model.

3.3.2 Creating a load

Different types of loads can be created:

- Nodal Loads
- Uniform Line Loads
- Varying Loads
- Uniform Surface Loads

The general UI for loads will be explained with the window for `Uniform Surface Loads` but is analogous for `Nodal Loads` and `Uniform Line Loads`. `Varying Loads`will be explained separately *here*.

After clicking on `Uniform Surface Loads` its window will appear.

Now follow these steps to create a new load:

1. Click on `New` to add a new load.
2. Select the newly created load, and change the name if needed.

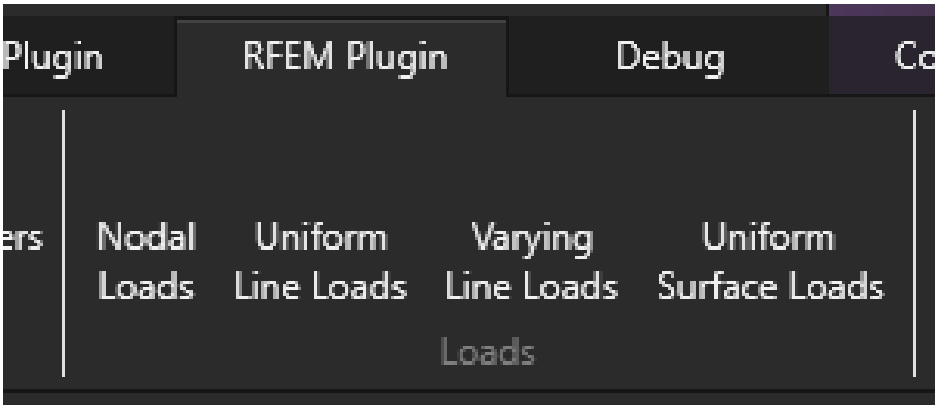

Fig. 3.6: Different types of loads that can be created with the plugin and its UI.

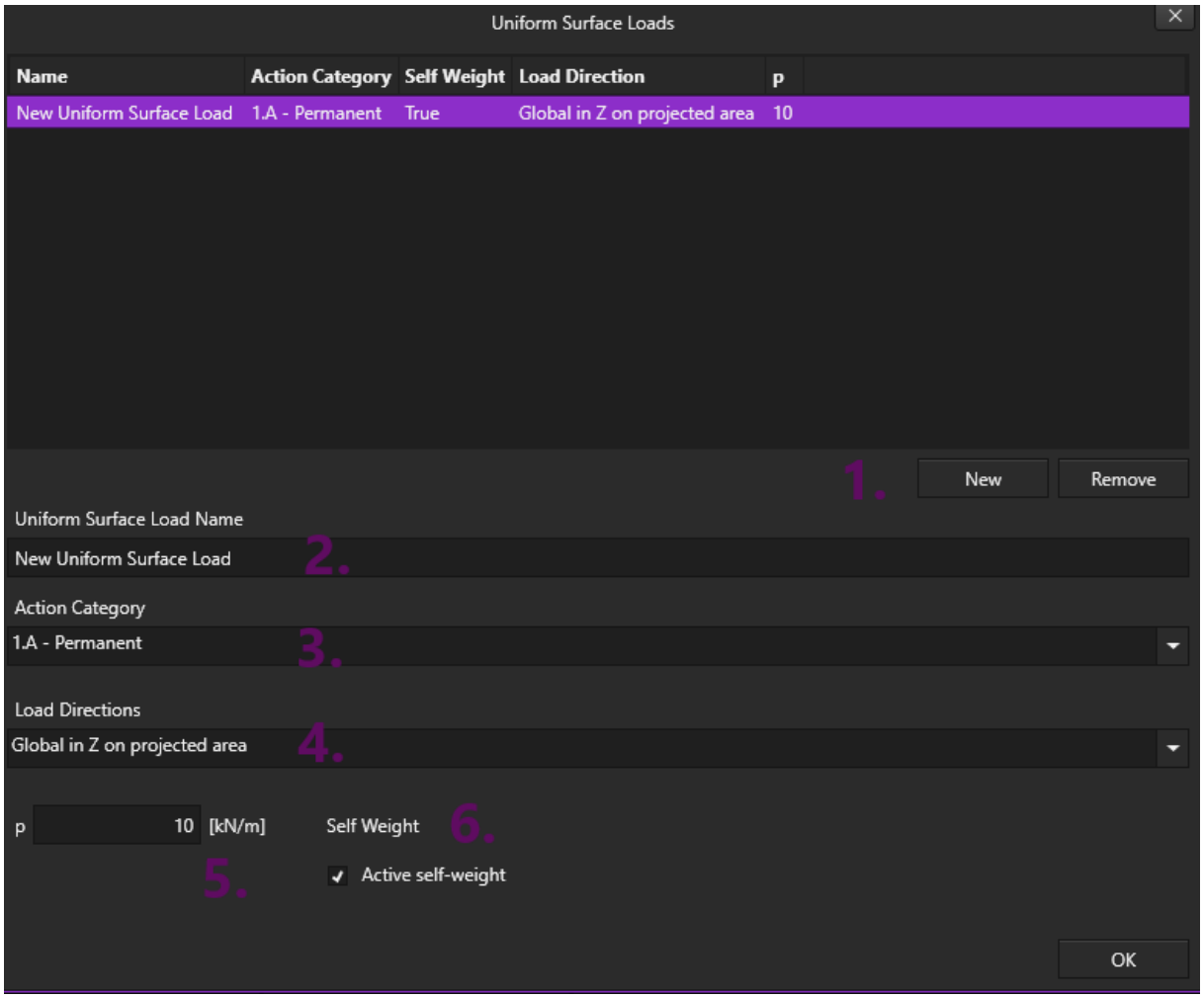

Fig. 3.7: UI for surface loads.

3. Choose the `Action Category`. These correspond to the ones in RFEM 6.
4. Choose the `Load Direction`. These correspond to the ones in RFEM 6.
5. Enter the magnitude of the load.
6. Check `Active self-weight` if needed.

Varying Loads

For `Varying Loads` the process is the same as described above, except that the magnitude of the loads have to be entered with a table.

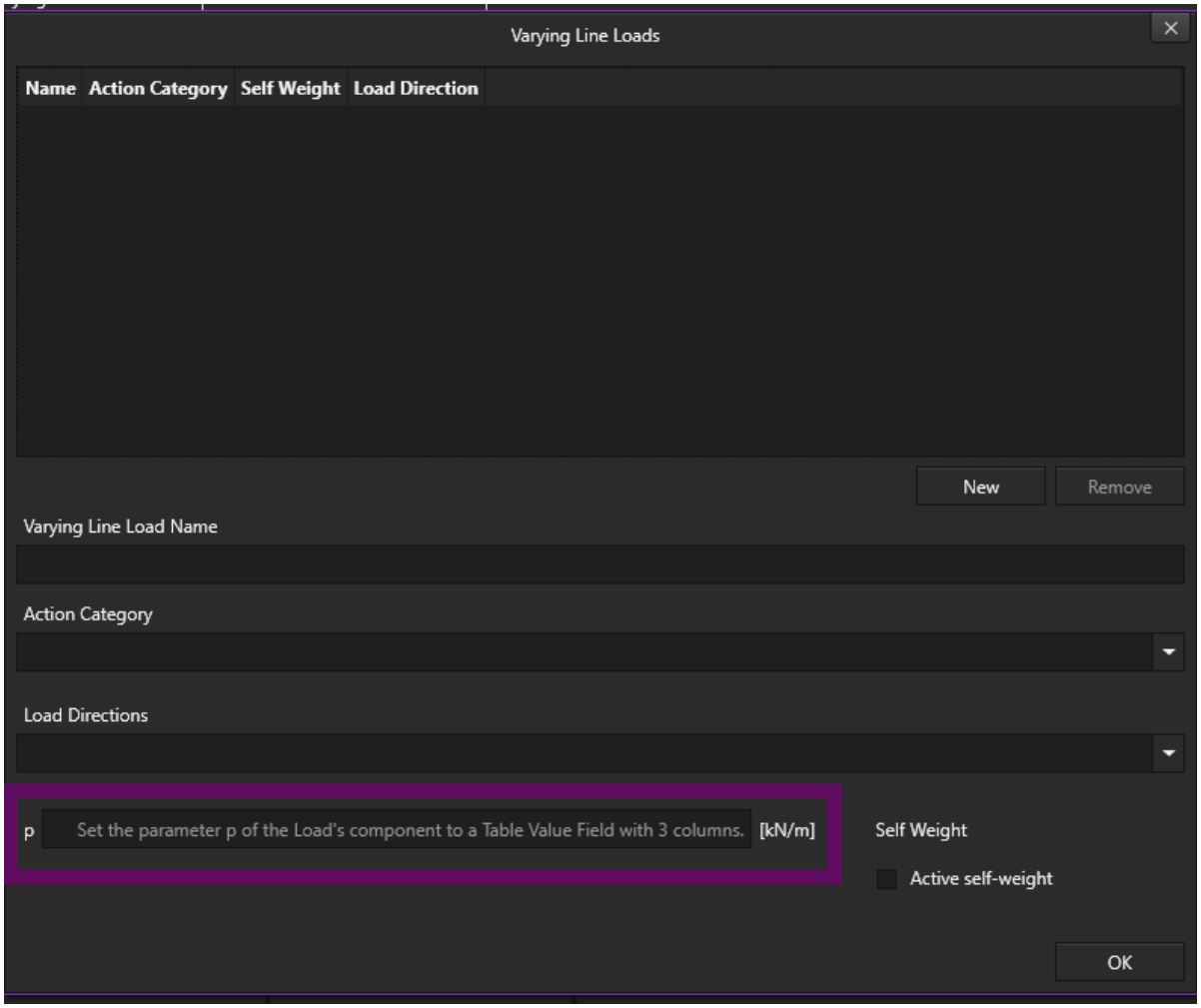

Fig. 3.8: UI for varying loads. Set the value in the table template created for the value by the plugin in the Editor.

After creating the load, the a template table to enter the magnitudes and their geometrical properties can be found under the created component in the editor.

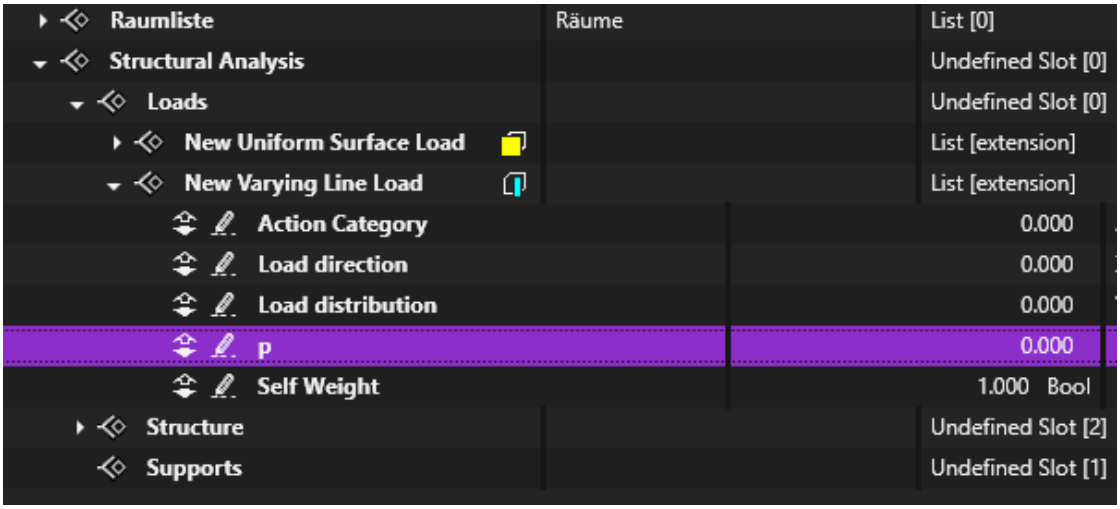

Fig. 3.9: Varying loads component in the `SIMULTAN-Editor`. Under the parameter `p` the template table can be found to adjust the loads.

3.3.3 Creating Supports

Different types of supports can be created:

- Nodal Supports
- Line Supports
- Surface Supports

Fig. 3.10: Different types of supports that can be created with the plugin and its UI.

The general UI for supports will be explained with the window for `Nodal Supports` but is mainly analogous for `Line Supports` and `Surface Supports`. Deviations will be explained separately and are mainly part of the `Nonlinearities`.

After clicking on `Nodal Supports` its window will appear.

Now follow these steps to create a new support:

1. Click on `New` to add a new support.
2. Select the newly created support, and change the name if needed.
3. Adjust `Translational Nodal Support Conditions`
4. Adjust `Rotational Nodal Support Conditions`

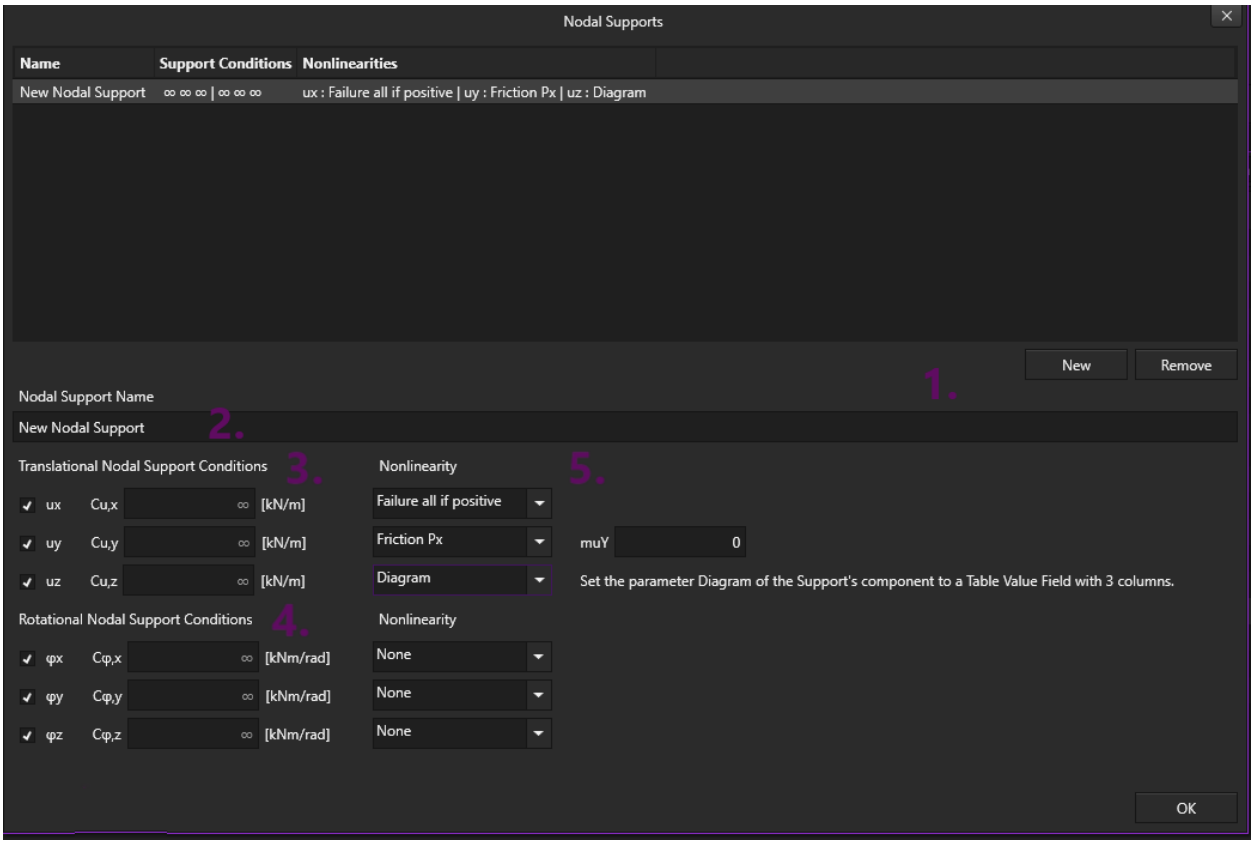

Fig. 3.11: UI for nodal supports.

5. Select `Nonlinearity` for the different directions if needed.

Note: Based on the `Nonlinearity` selected either value fields will appear or there will be a note to adjust the values in the table of the created component if needed.

3.4 Connecting components with the geometry

The before created components are a representation of the information needed for the structural analysis. Connect the components to the geometry as needed. Please consult the SIMULTAN Editor User Guide for general information on how to link components and geometrical information.

An outline of the most important links and how to create them will be given here.

3.4.1 Surfaces

Information for surfaces are autogenerated via the already existing reference geometry since this data is not exclusive to structural analysis.

3.4.2 Members

Geometrical lines connected to physical properties for structural analysis, e.g. beams or columns, are called members in RFEM. To add a `Section-Component` existing in the data model to a line follow these steps:

- select the line in the `Geometry Editor`
- Open the `Property Editor` for the selected line
- Choose the Component you want to link from the pop-up window

3.4.3 Supports

Supports are linked to vertices/lines/surfaces in the geometrical model. To add an existing `Support-Component` to a geometry in the geometrical model follow these steps:

- Select the node/line/surface in the `Geometry Editor`
- Open the `Property Editor` for the selected node
- Choose the Component you want to link from the pop-up window

3.4.4 Loads

Loads are linked to vertices/lines/surfaces in the geometrical model. To add an existing `Load-Component` to a geometry in the geometrical model follow these steps:

- Select the node/line/surface in the `Geometry Editor`
- Open the `Property Editor` for the selected node
- Choose the Component you want to link from the pop-up window

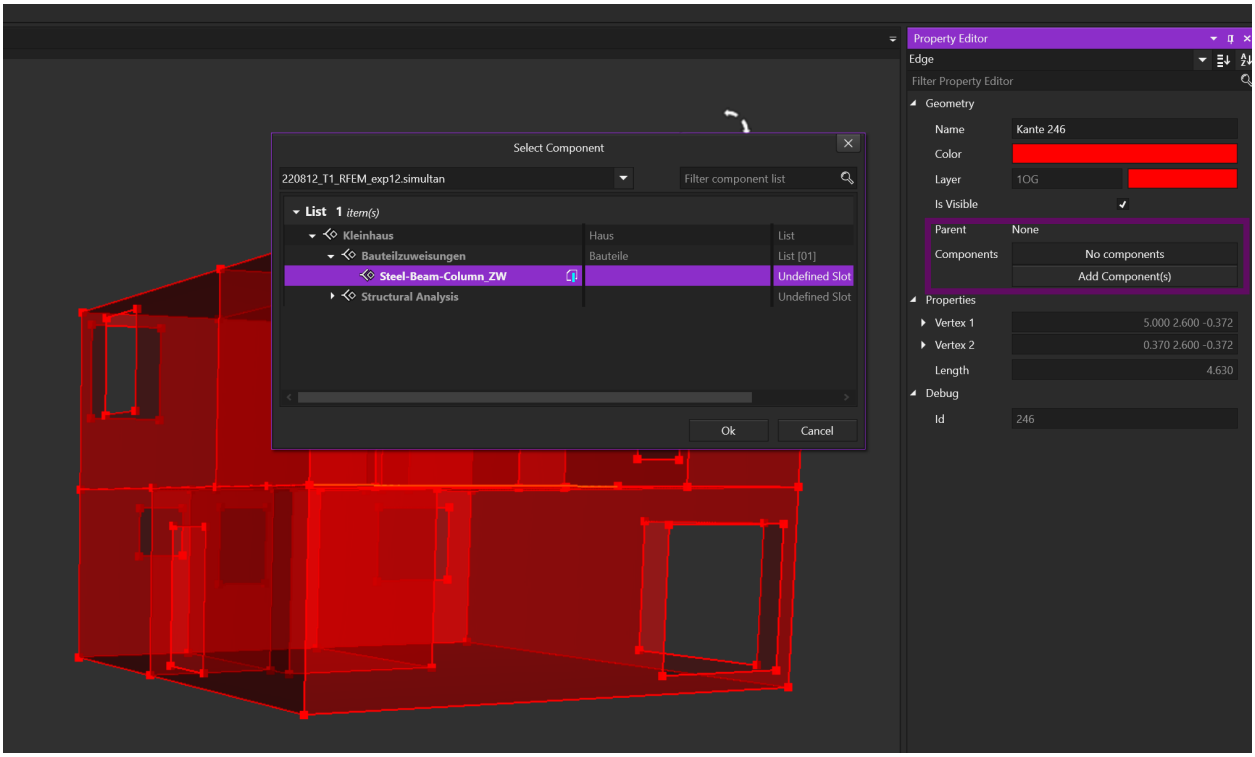

Fig. 3.12: Example how to link a `Section-Component` to a geometrical line in the `Geometry Editor` using its `Property Editor`.

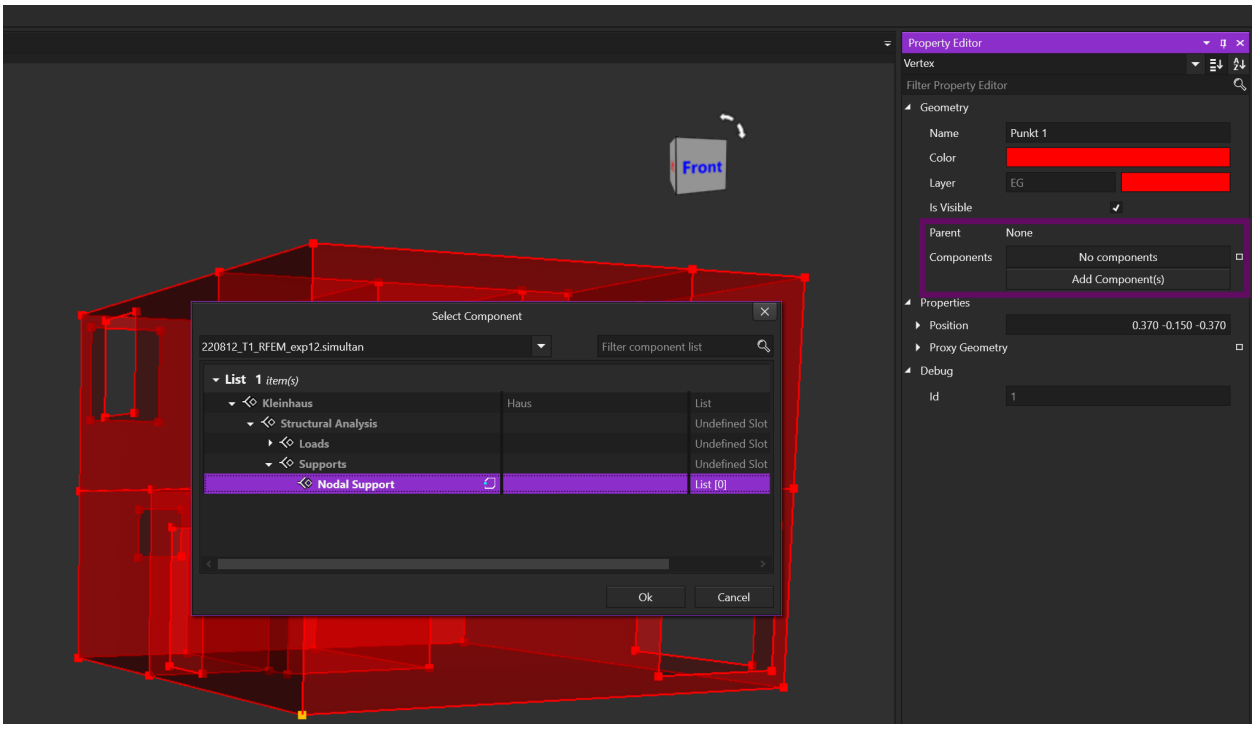

Fig. 3.13: Example how to link a `Support-Component` to a geometrical node in the `Geometry Editor` using its `Property Editor`.

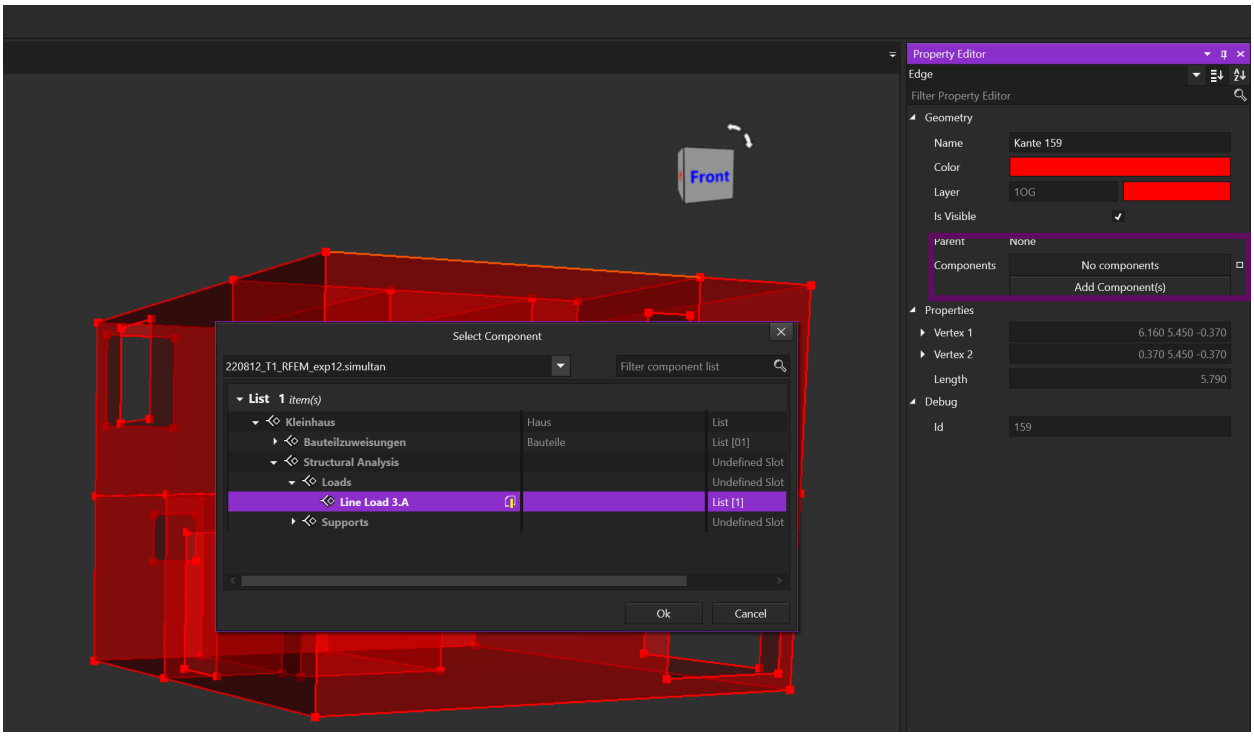

Fig. 3.14: Example how to link a `Load-Component` to a geometrical node in the `Geometry Editor` using its `Property Editor`.

Note: The same `Support-Component` or `Load-Component` can be linked to multiple different geometrical instances. For example, if the same force is attacking on two differnet surfaces, the same component is used from the data model. An in depth look at this you can find in *this chapter*.

CHAPTER

FOUR

RUNNING A SIMULATION

4.1 Exporting the SIMULTAN-model to RFEM

When modelling according to the conventions described in chapter *Setting up a problem* the model is ready to be exported. This can be done by clicking the `Export Model` button when you are inside the RFEM Plugin in the SIMULTAN Editor Fig. 4.1.

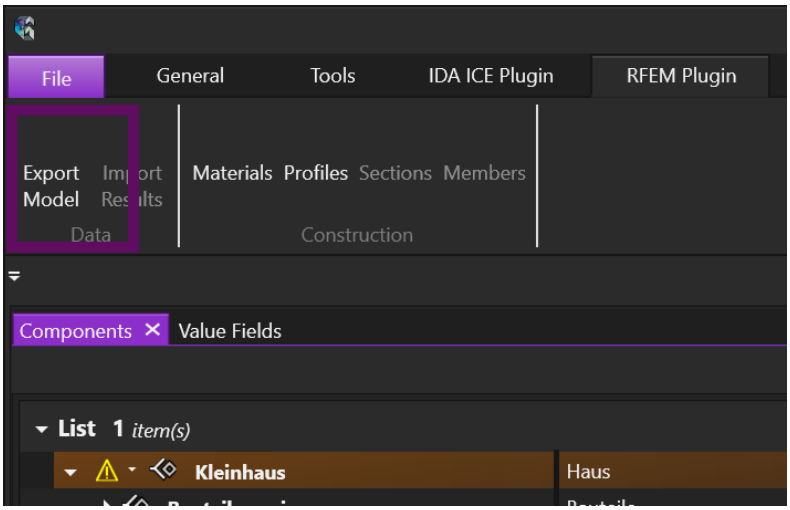

Fig. 4.1: Export button to create the XML-file to import into RFEM 6.

After clicking the button you will be prompted with a `save dialogue`. Please specify the location where you want to save the XML-file and confirm.

4.2 Importing into RFEM

To import the XML-file open RFEM 6 and choose the `Import XML function` Fig. 4.2.

Navigate to the location on your disk where you have saved the exported model and confirm to load it. After the model is loaded the it will appear in RFEM 6 and is ready for further manipulation Fig. 4.3.

Warning: In the current state of development the plugin is **NOT** bidirectional. Any adaptations done in RFEM are only available in RFEM and cannot be retrieved into the SIMULTAN datamodel. This is a funcitonality which will be implemented in the future. For now use RFEM as a solver and adjust the model in the SIMULTAN Editor.

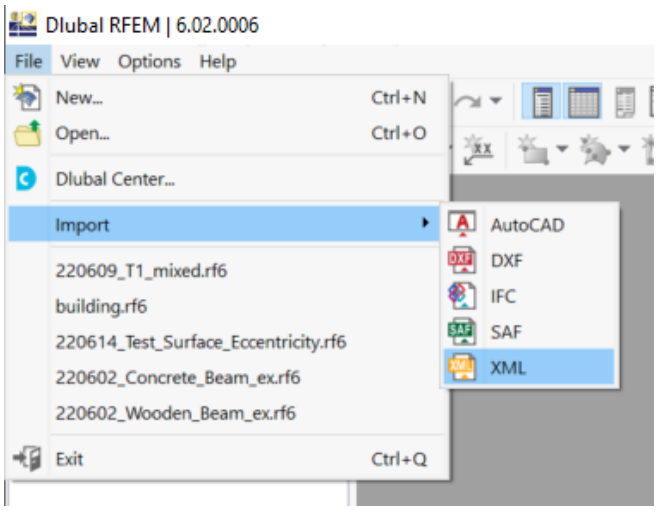

Fig. 4.2: Import XML function in RFEM 6.

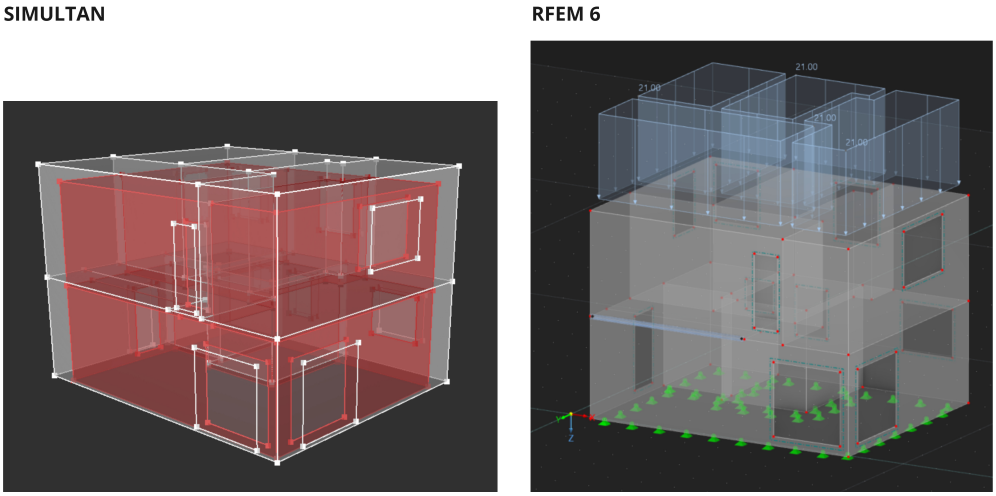

Fig. 4.3: Imported model in RFEM 6 based on an export from a SIMULTAN model.

4.3 Running a simulation in RFEM

To run a simulation create load combinations in RFEM and run the simulation as usual in the finite element software.

CHAPTER

FIVE

RESULTS OF THE SIMULATION

5.1 Visual results

For now the simulation results can only be viewed in RFEM itself, Fig. 5.1.

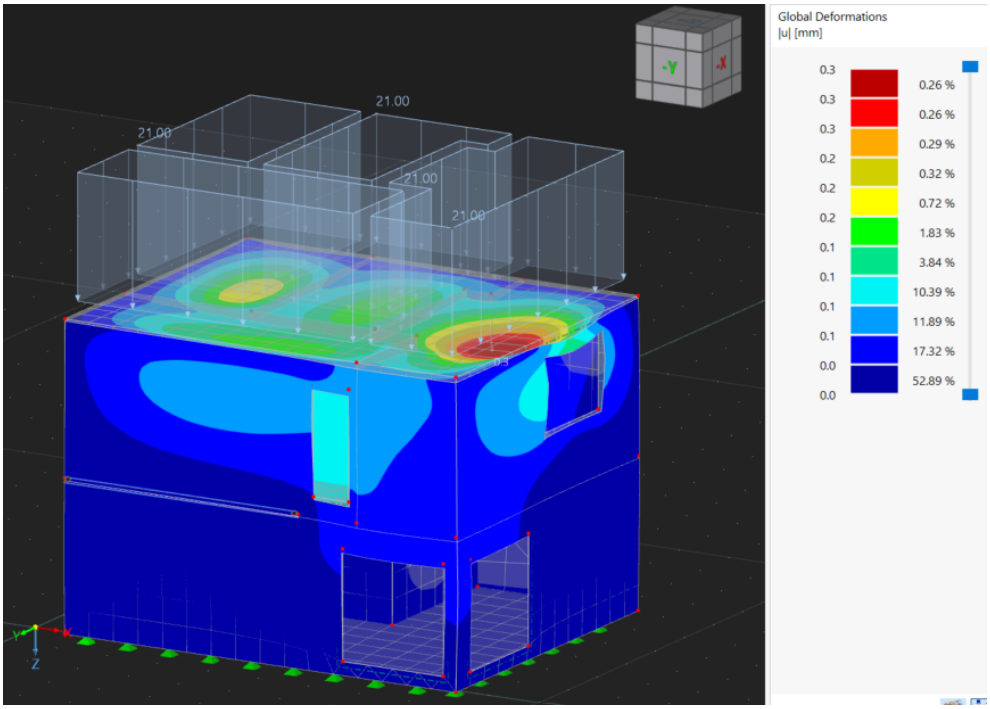

Fig. 5.1: Results of a FE-analysis with RFEM 6 for an exported SIMULTAN model.

5.2 Additional information

This restriction is due to the fact that the SIMULTAN Editor has no functionality for mesh visualization implemented yet. Since simulation results of finite element calculations are dependent on proper visualization it was decided to delay this feature till the SIMULTAN Editor can provide this functionality.

Note: As a workaround one can attach the calculation protocol and the RFEM model to the `Structural Analysis Component`.

CHAPTER

SIX

SIMULTAN DATASTRUCTURE TO INCORPORATE THE RFEM DATA MODEL

6.1 Digital Twin and data model for structural analysis

In Fig. 6.1 a simplified depiction of the digital twin and the `Structural Analysis Component` is given. This component is used in the SIMULTAN datamodel to store the information needed to export data with the RFEM Plugin to enable structural analysis simulations in RFEM 6. The global structural analysis can be simplified into 3 main containers of information, which are independent of each other. They are represented by their own compoenents in the SIMULTAN data model, everyone of them holding sub-components for further detailed information.

The connection and dependency between some of these components is indicated in Fig. 6.1 by the blue lines with arrows next to `Component with physical information`, `Component for geometric assignment` and the `Structural Analysis Component`.

In the SIMULTAN Editor these components could be represented in the ways shown in Fig. 6.2 and Fig. 6.3.

6.2 Structural Analysis Component in depth

If we take a more detailed look into the `Structural Analysis Component` we can see the references and dependencies created to link and store information in Fig. 6.4. Here the data structure with modelling information, references and dependencies for a section are shown. In this case this section represents a steel beam which will be exported to RFEM 6 with the plugin for a finite element analysis.

6.3 Data modelling decisions for data representation

Due to the fact that in structural analysis often times the same data is used in different geometrical locations (e.g. force attacking at multiple locations) the component storing the information for these repeatedly used data is only created once and instances of the component are storing the differing geomtrical information. This helps to avoid cluttering of the data model is avoided.

The instances can be accessed from the SIMULTAN Editor as shown in Fig. 6.5 and further information is displayed in the opened tab, Fig. 6.6.

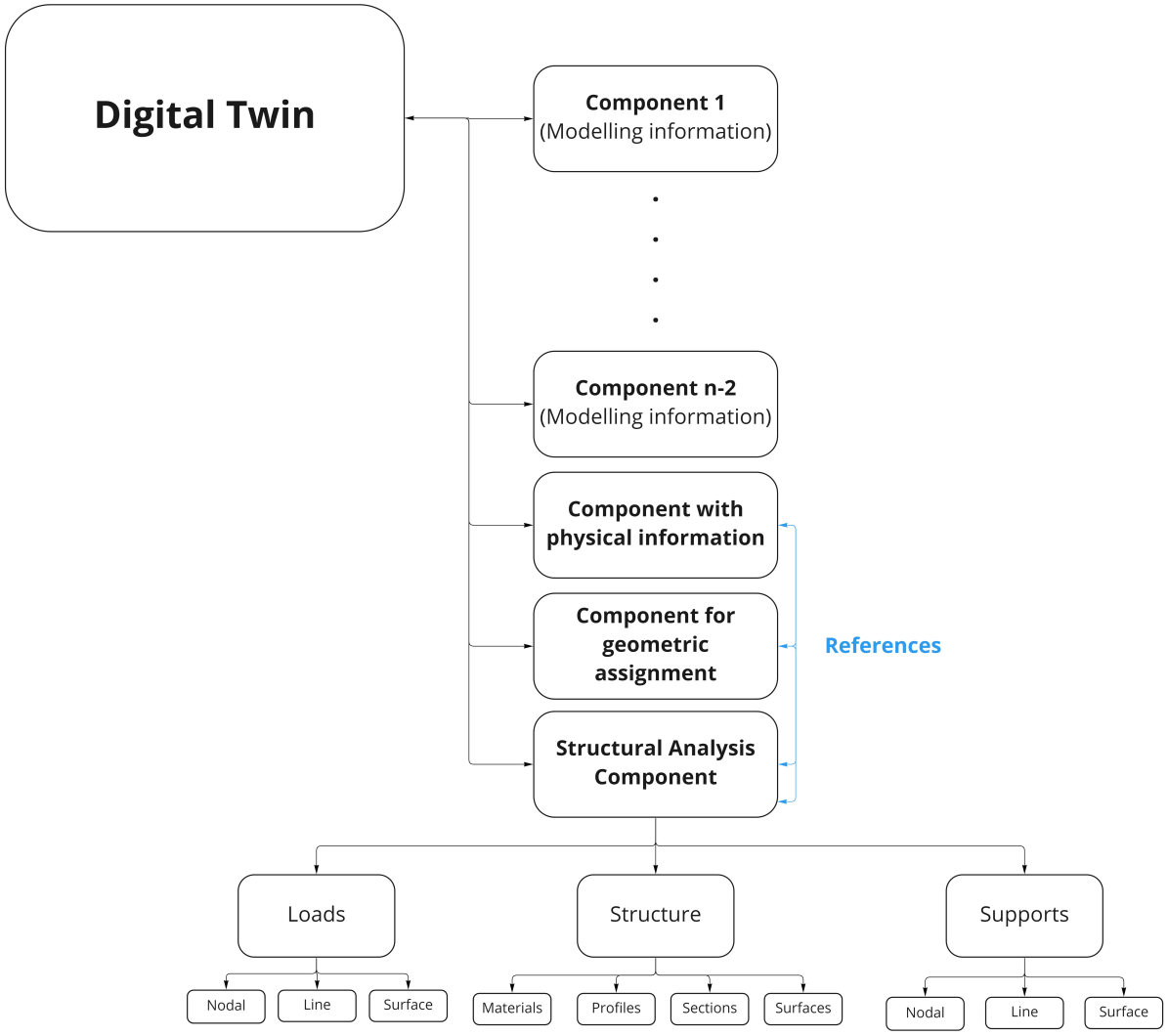

Fig. 6.1: Simplified depiction of the representation of a Digital Twin with a `Structural Analysis Component`.

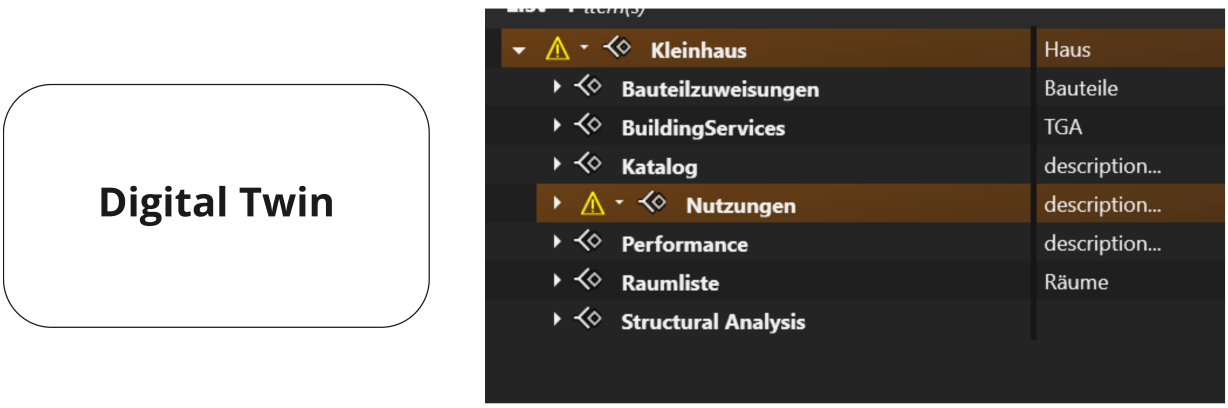

Fig. 6.2: Collapsed model of a digital twin for a *Tiny House* in the SIMULTAN Editor.

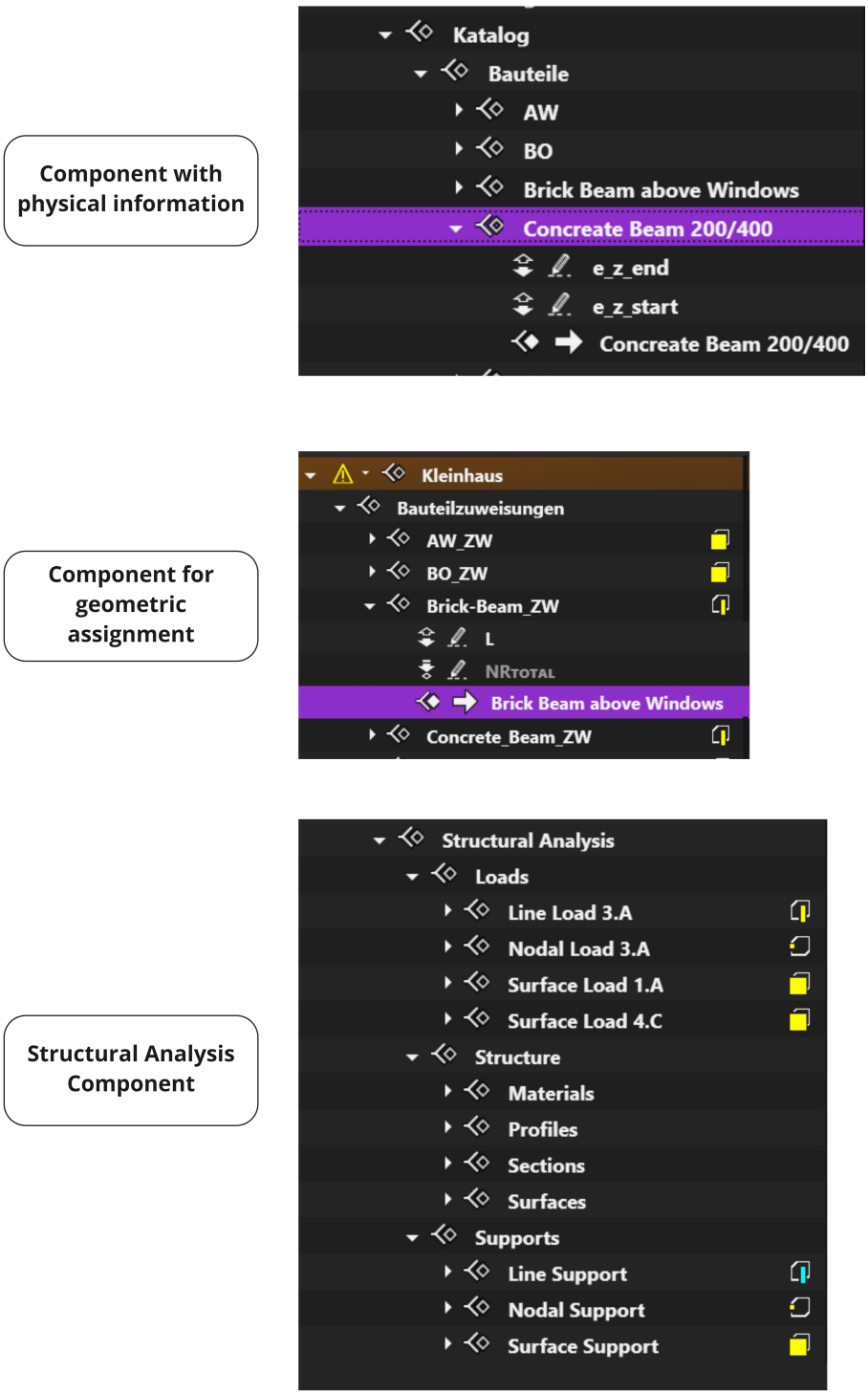

Fig. 6.3: Representation of the depicted components in the SIMULTAN Editor.

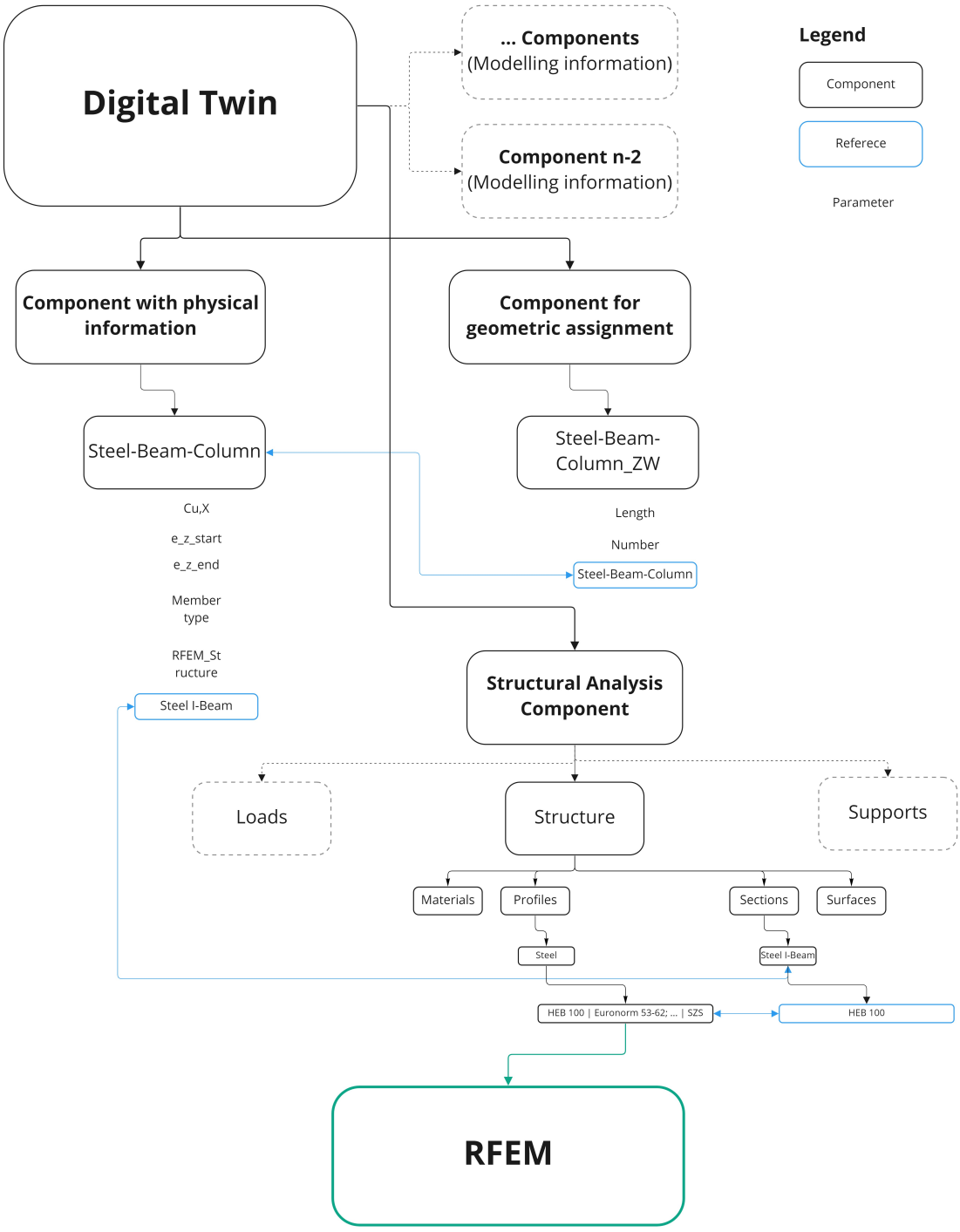

Fig. 6.4: Detailed depiction of the representation of a section in the Digital Twin.

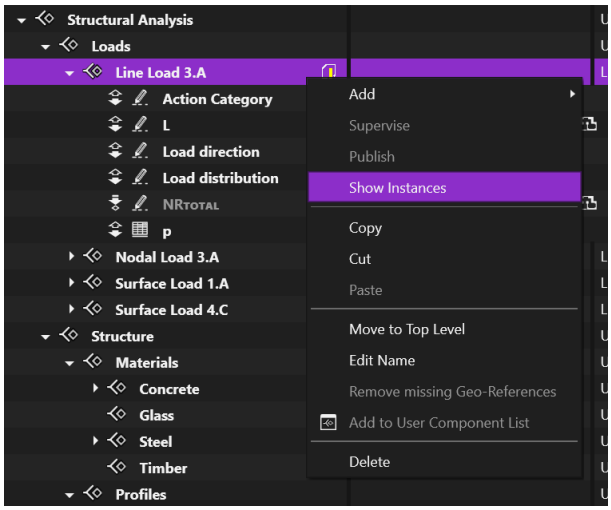

Fig. 6.5: Right click to access instances of a component.

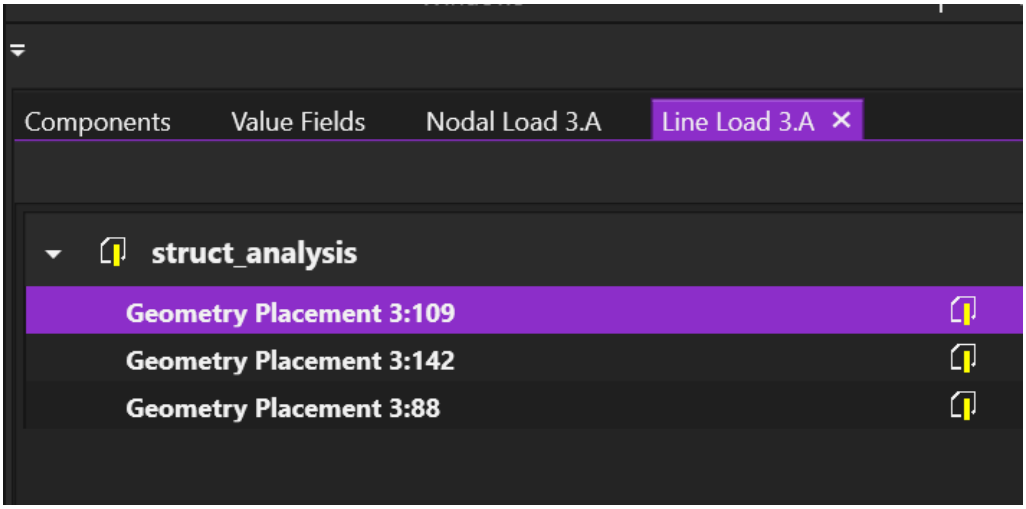

Fig. 6.6: Further information can be accessed from the opened instances tab.

CHAPTER

SEVEN

COPYRIGHT AND LICENSE AGREEMENTS

Please reference this repo when including information or knowledge provided here in your work.

CHAPTER

EIGHT

REFERENCES

BIBLIOGRAPHY

[Betal] **missing year in bednarSIMULTANSimultanePlanungsumgebung**

[BuhlerSB22] Maximilian Bühler, Bernhard Steiner, and Thomas Bednar. Digital Twin applications using the SIMULTAN data model and Python. In *WBC 2022*. Melbourne, 2022.

[PLW+19] Galina Paskaleva, Thomas Lewis, Sabine Wolny, Bernhard Steiner, and Thomas Bednar. SIMULTAN as a Big-Open-Real-BIM Data Model - Proof of Concept for the Design Phase. In *CIB World Building Congress 2019 Constructing Smart Cities*, 10. 2019.