



UNIT 01 | INTRODUCTION

The traditional methods, established in the various Standards, continue to over-simplify, with the aim of providing a solution to as many of the different configurations and types of connections as possible. Consequently, the numerous possibilities for combining elements end up being a common cause of uncertainty and insecurity due to insufficient knowledge about how to check the connection correctly.

Today there exists a considerable number of applications related to structures analysis and specifically to the calculation of connections for steel structures. Nevertheless, this phase of the structural project continues to be a problematic step that many engineers would prefer to avoid. The fault lies with the existing software which, based on traditional methods, contains a series of limitations that prevent the analysis of all possible cases, some of which are not even as complex as one might think.

The connection is simple: a beam to column joint. However, the beam has internal forces applied to it in the 6 degrees of freedom (axial, vertical shear, horizontal shear, torsional moment, vertical moment and horizontal moment). How do we calculate the internal forces to check the fillet weld?

What do we do with spatial connections? These aren't unusual: a corner connection in an industrial hangar or in a rack are common examples. They are usually calculated in a simplified way, element by element, but what happens with the interaction of the internal forces? How do we know if it's necessary to reinforce the support or not?

What about connections of cold-formed cross-sections? It's a manufacturing process that is starting to be widely used for the possibilities it offers for temporary or auxiliary structures.

The search for answers, to these and many other questions, has been behind the development of IDEA StatiCa Connection, software based on analysis using the Finite Element Method.

Contrary to popular belief, this type of analysis is nothing new, since the first numerical models and simulations were built over four decades ago. The reason it has not been used until a few years ago is that the computational power of the computers was insufficient to solve the numerous equations generated by this type of numerical methods.

It therefore makes no sense at all that, with the computational power that exists today, we are limited by our tools. With this philosophy, IDEA StatiCa Connection proposes that limitations should only arise from the particular circumstances of the project or, indeed, the imagination of the designer, given that anything that can be modeled can be calculated by the program.

Thus, the team at Construsoft's Department of Analysis and Design welcomes you to this course. In addition to learning how to use the program, you'll see some basic notions of the analysis method used, as well as the associated check procedure that, of course, has long been validated and recognized by the various Standards.

Barcelona, July 12th, 2019.

Analysis and Design Department of Construsoft

01.01. GENERAL INTERFACE AND WORKFLOW OF THE PROGRAM.

As you can see in the following image, the program interface is divided into 4 large modules, of which 2 will be completely static throughout the modeling process and 2 will be variable depending on the tool used. Each of these modules is discussed in detail below.

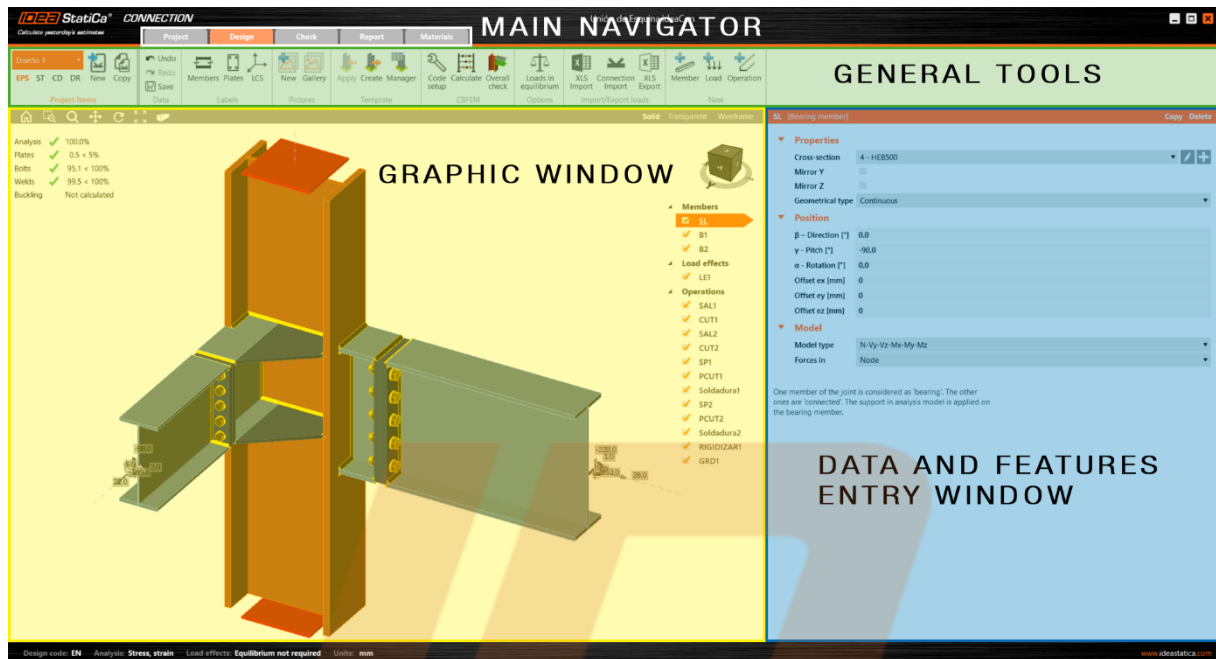


Figure 01.1. Appearance and general layout of the program interface.

MAIN NAVIGATOR

The main navigator module will be the vehicle through which the entire modeling process is managed. It represents the program's structure and mode of operation, as well as showing which work phase, in the analysis of the connection, is being used at any given time.

This navigator has five tabs, as described below. Of these, those corresponding to *Design* and *Check* have special importance, since they will be the most frequently used.



Figure 01.2. Main navigator tabs

PROJECT. In this section you can determine the different general parameters, the display of the program, and that of the project, in terms of the organization of the report.

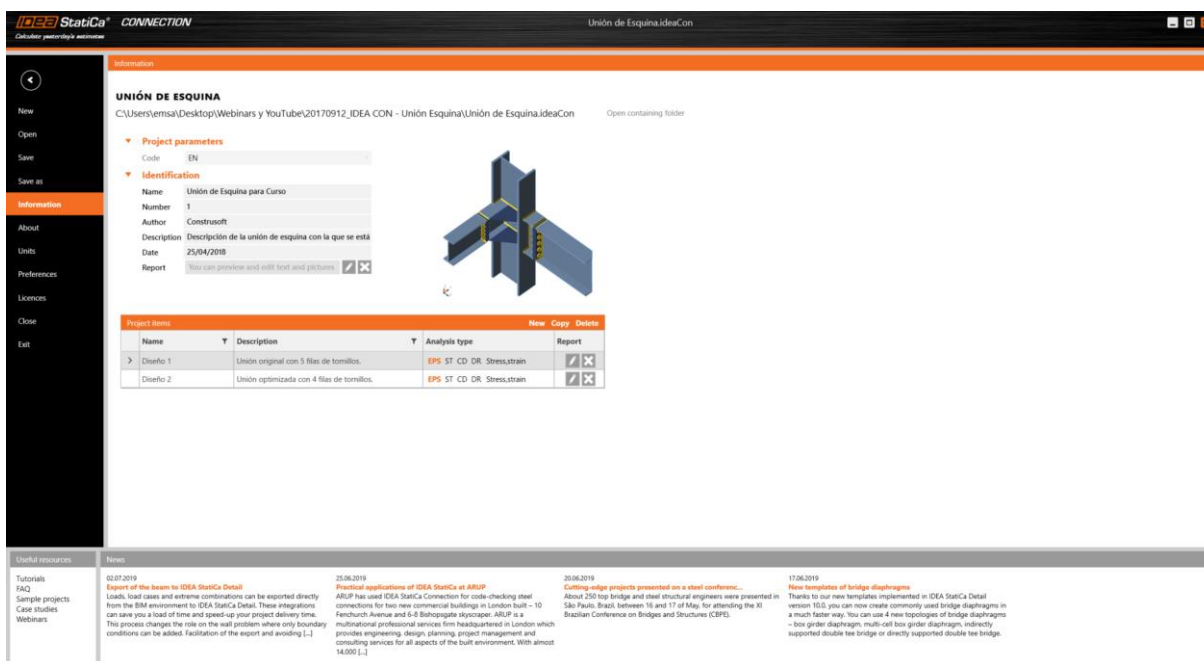


Figure 01.3. General appearance of the *Project* window.

As you can see, it consists of a side navigator (on the left) through which you can access the different options. In the following chapters, the most relevant sections of this navigator will be discussed in more detail.

In the central window, all the information related to the file that has been opened is displayed, from the path showing where the file is stored to specific parameters of the project that can you can configure so as to include them in the connection report and documentation.

In addition, you can also find a manager of the different connections, or proposals, that have been made within the file. In other words, IDEA StatiCa allows different connections to be stored within the same file so that a quick comparison can be made between different design proposals, different types of analysis, etc.

Finally, in the bottom section you can find various web links with the latest news published on the official IDEA StatiCa website, as well as other links of interest to digital resources that facilitate the resolution of problems and queries.



Using the icon in the top left corner, you can return to the previous window of the program which, by default, is the *Design* tab.

DESIGN. As its name suggests, all the modeling of the connection to be evaluated will be carried out here. This design process consists, in turn, of 3 subcategories: Member, Load and Operation.

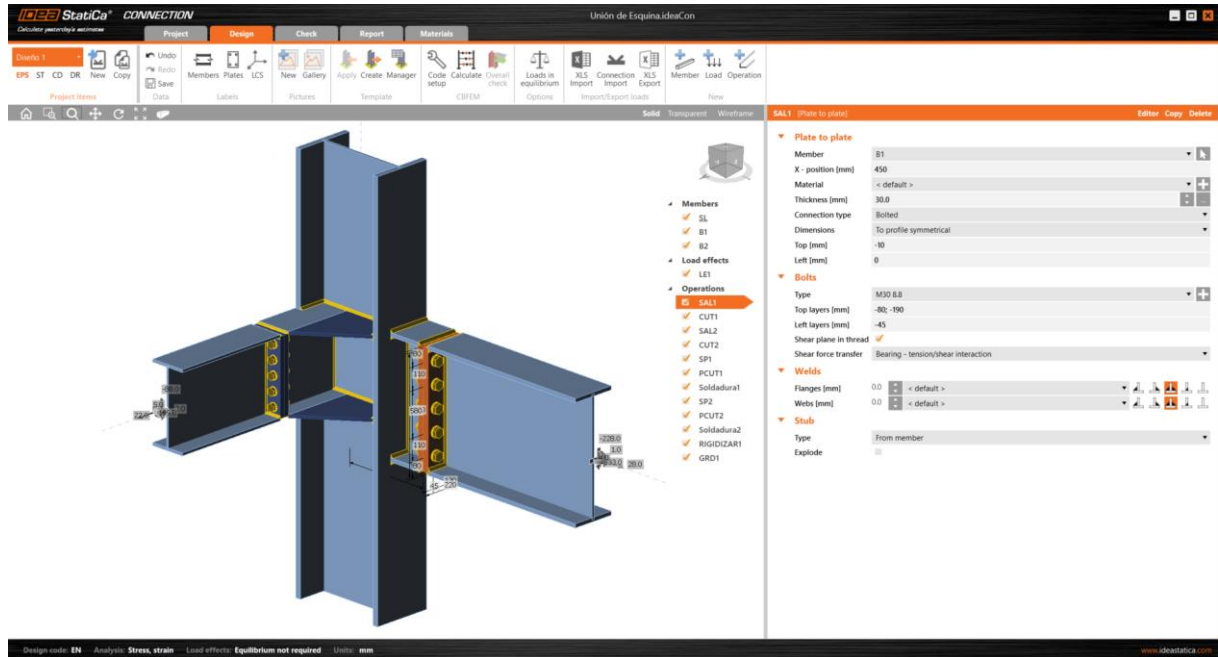


Figure 01.4. General appearance of the *Design* window.

In the *Member* section, the main elements that make up the connection: columns, beams, braces, etc., will be available, i.e. all those elements on which loads will be applied.

In the *Load* section, as might be expected, the different combinations and their corresponding forces, with which the connection will be evaluated, will be introduced.

Finally, in the *Operation* section each of the actions necessary to connect the various elements that have been introduced previously will be defined.

CHECK. In this window, the connection check will be carried out, as well as the visualization of all the possible results that can be obtained.

As can be seen in Figure 01.5, the layout of this window is very similar to the one seen on the Design tab, being composed of the different modules that are explained further on in this same introductory unit.

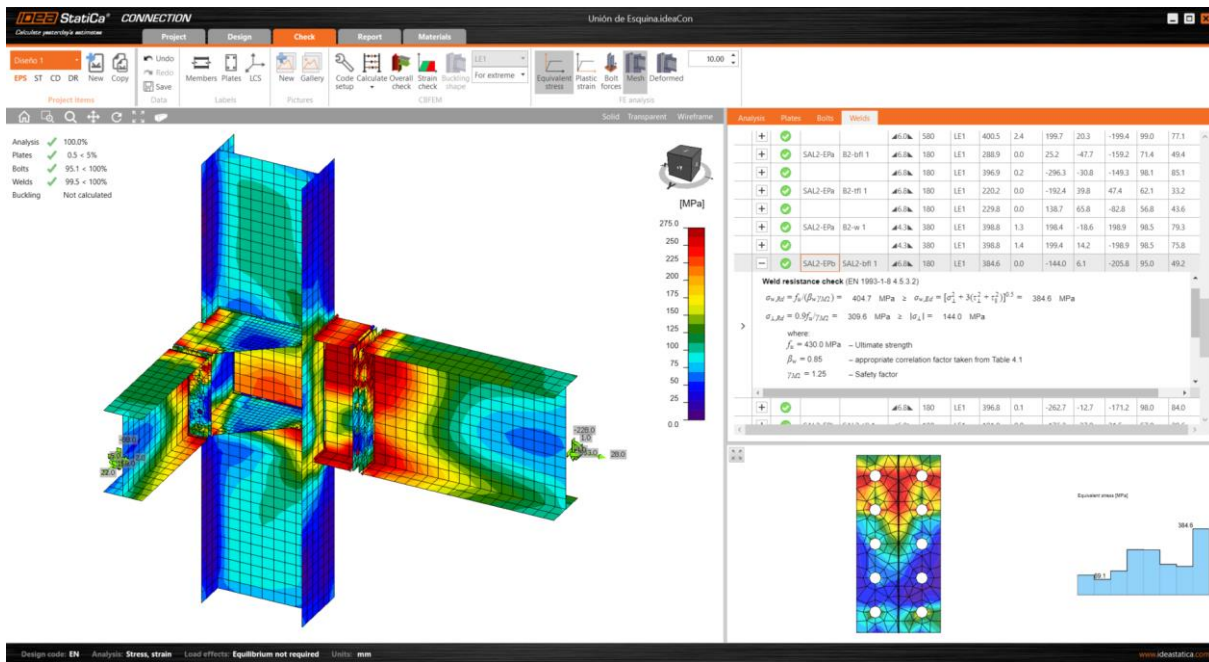


Figure 01.5. General appearance of the *Check* tab.

REPORT. Once the modeling and analysis of the connection is complete, the last step of the workflow is the generation of the Report that will be included in the project's analysis documentation. On the *Report* tab, the sections to include in the documentation can be specified.

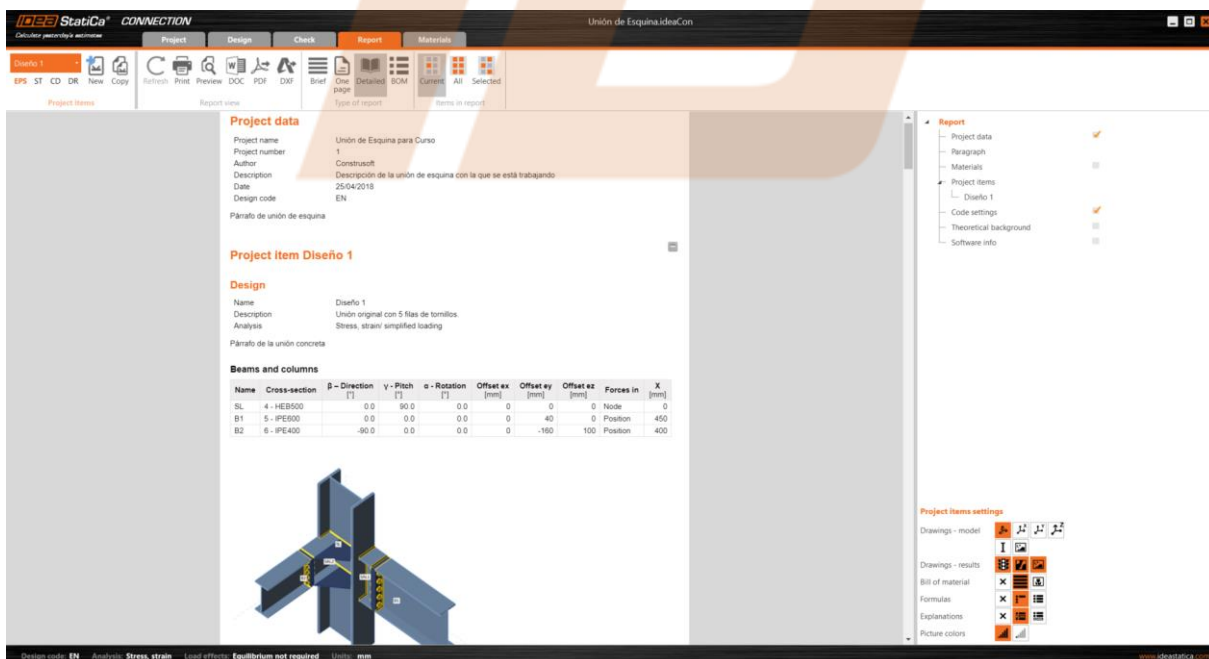


Figure 01.6. General appearance of the *Report* tab.

In addition, from this tab you will be able to access specific, detailed information about the different elements that make up the connection, as well as manufacturing drawings, with dimension lines, that can be printed and exported to a .dxf format.

MATERIALS. In this section you can view and edit all the sectional and material parameters that are used in the project, as will be explained in chapter 01.03.

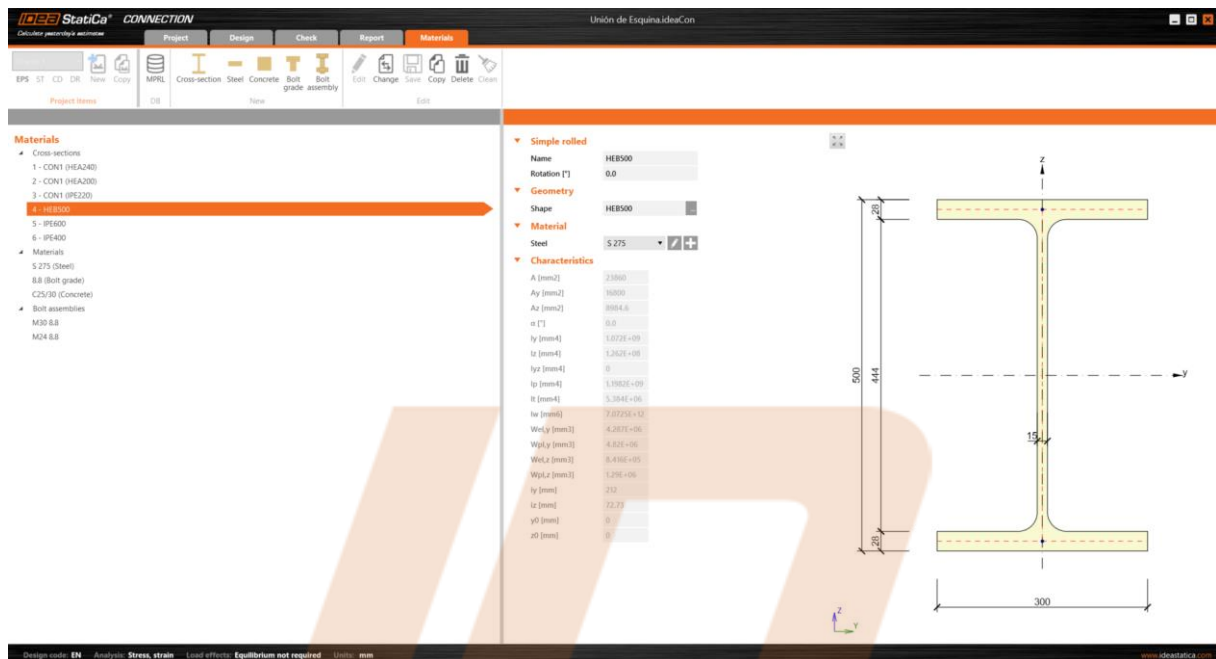


Figure 01.7. General appearance of the *Materials* tab.

GENERAL TOOLS

In this module, there are various sets of tools related to visual and working aspects of the model. Depending on the Navigator tab that is being used, different sets of tools will be available, as shown in Figure 01.8.

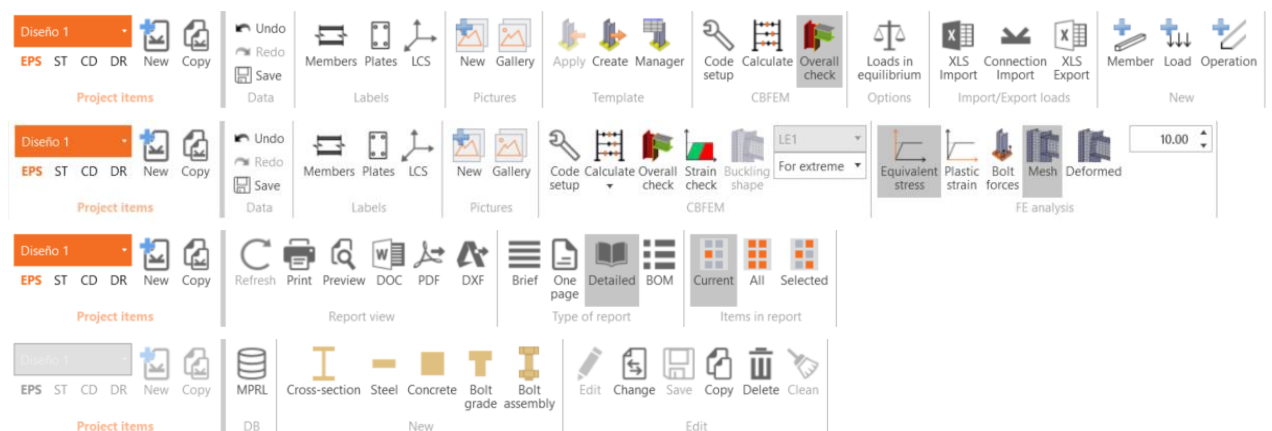


Figure 01.8. Configurations of the General Tools module according to the modeling phase.
From top to bottom: Design, Check, Report and Materials.

The specific tools of each tab will be explained in the teaching units related to each phase. In this section those tools that remain constant throughout the process will be described.



Figure 01.9. Basic work tools.

Project items. First, an item (connections) selector appears. As previously mentioned, it is possible to have several connection models within the same file. In this area you can quickly switch between models, modify the type of analysis performed, as well as create new models and duplicate existing ones. In this way, carrying out comparisons is much more direct.

Data. Classic *Undo/Redo* tools, as well as *Save*.

Labels. As its name suggests, with these tools you can activate or deactivate the display of labels or reference objects on the model in the graphic window.

Through the LCS icon you can activate the visualization of the local axes of the elements, which will serve as a reference to position the elements, as well as for the introduction of loads on them.

Pictures. In addition to the multiple options offered by the program for configuring the report, you can also create a set of images of the connection, determined by the user and stored in the Gallery, to be attached to the report. The possibilities and operation of this tool are explored in greater depth in the Reports and Documentation teaching unit.

GRAPHIC WINDOW

The primary advantage of this window is that it provides additional information (graphics) during the modeling, enabling easier and more intuitive modeling. In addition, it's a way of verifying that the features of the connection components are being defined correctly.

Furthermore, it has a field that provides specific information about the selected element, which will be useful frequently during modeling.

In version 9 of the program, this window takes on a much larger role, becoming the center of reference around which the whole process revolves.

Much more information has been added, while other menus have been simplified. A new, more graphic and intuitive work system, using the right mouse button, has been implemented. Then, other graphic tools have been added, as shown in the image below:

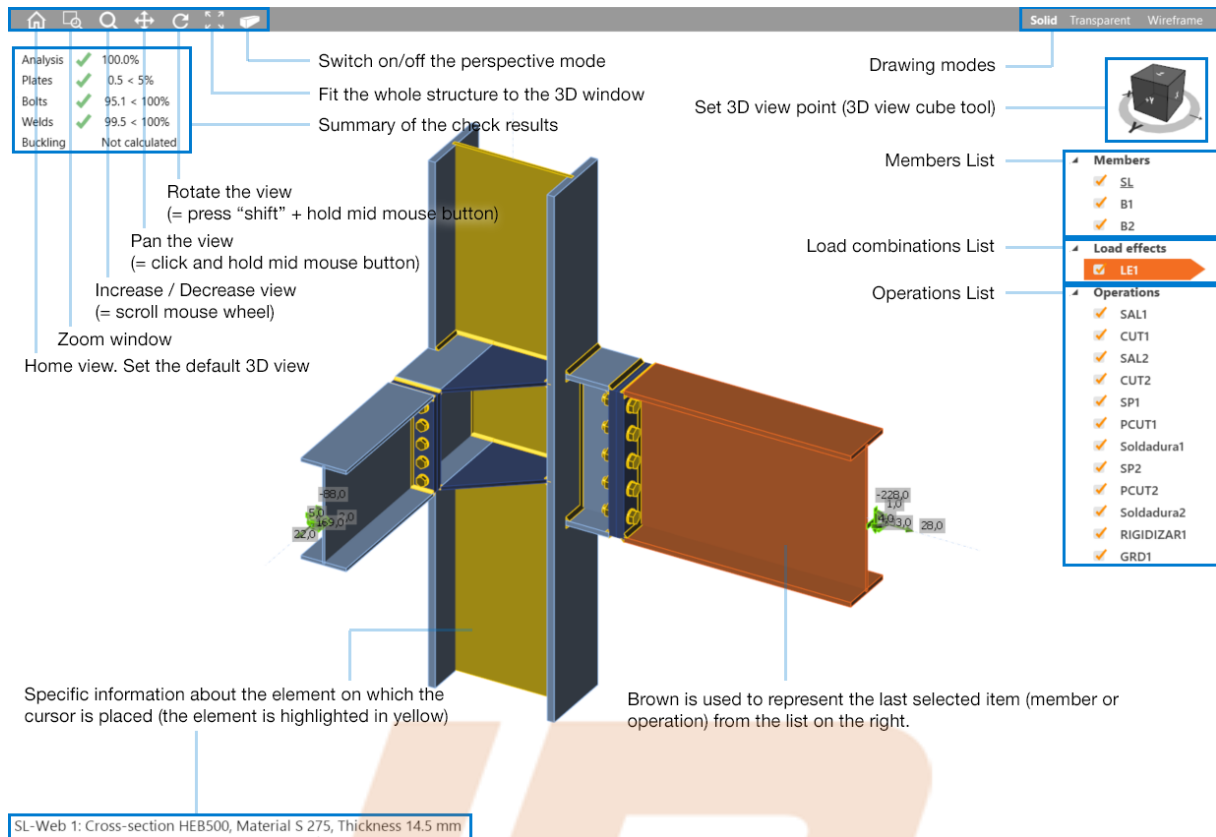


Figure 01.10. Appearance and description of the graphic window.

Drawing modes. With these tools we control the basic appearance of the image displayed in the program's graphic window.

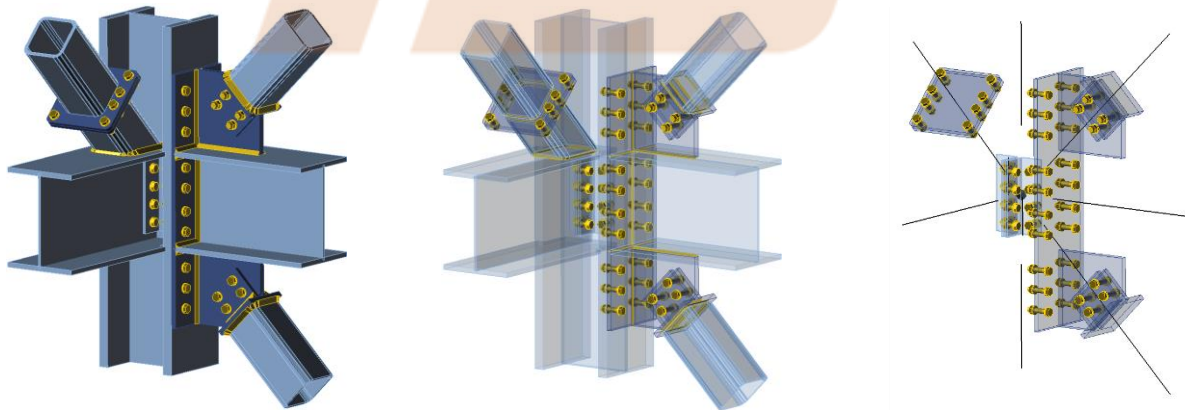


Figure 01.11. Configurations of the view of the connection. Solid, transparent and wireframe.

You can choose how to visualize the model, from the solid view to the wireframe diagram. The transparent display can be useful for the positioning of elements in complex joints, in which the visualization of certain parts can be difficult. As for the wireframe diagram, it has special importance in the introduction of loads on the node, as will be seen later.

DATA AND FEATURES ENTRY WINDOW

This is one of the variable modules of the program, the appearance and features of which will depend on the item that has been selected in the Graphic Window list.

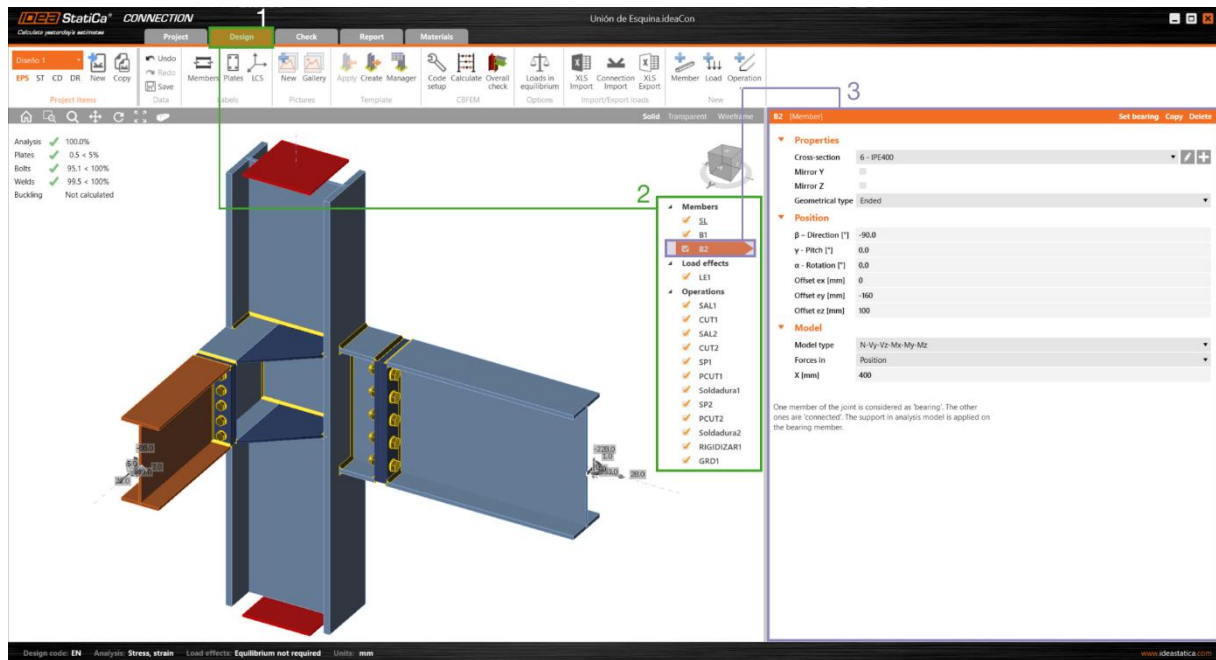
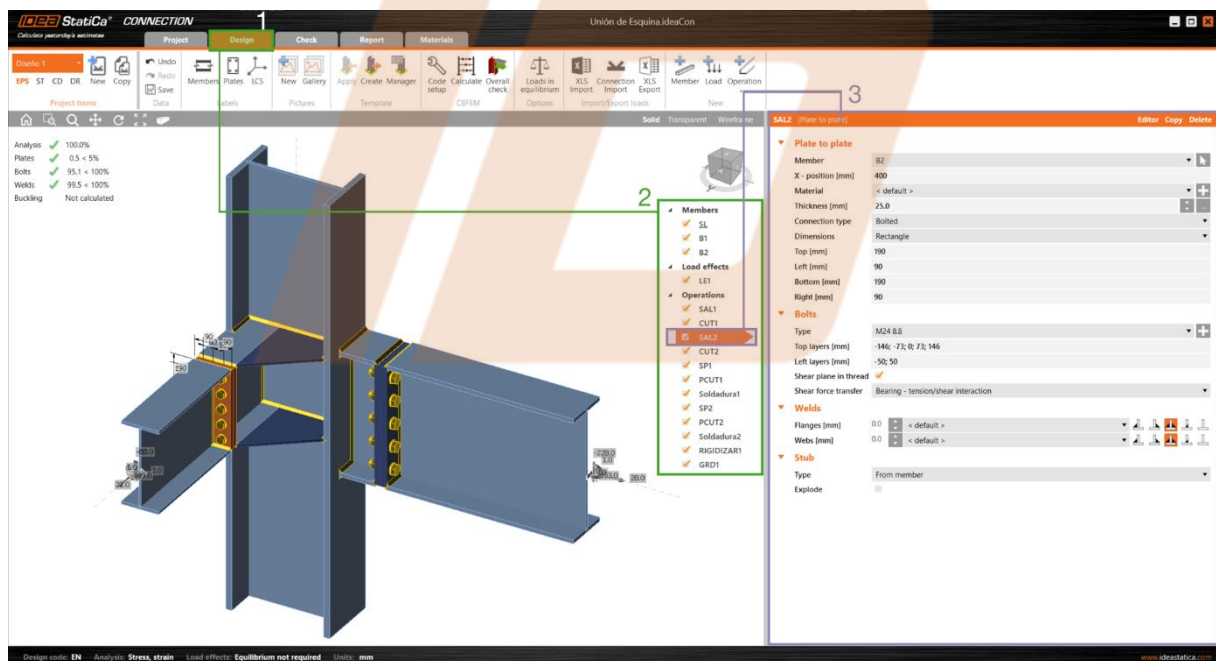


Figure 01.12. General appearance of the Data and Features Entry tab.

Although this module will depend on the selected item, it will usually present a similar structure, grouping the different property fields according to typology (geometric properties, components of the operation, position definition, etc.).

In the header of this module are the quick tools to duplicate operations and access the advanced geometry editor, as will be seen in the Teaching Unit 04.

Therefore, the workflow with IDEA Connection will have the same hierarchical scheme as that shown, as an example, in the images below:


 Figure 01.13. Workflow example with the *Member* item.

 Figure 01.14. Workflow example with the *Operation* item.