User manual
CYPE-Connect

CYPE-Connect

User manual

Software for
Architecture,
Engineering and
Construction

CYPE Ingenieros, S.A.
Avda. Eusebio Sempere, 5
03003 **Alicante**
Tel. (+34) 965 92 25 50
Fax (+34) 965 12 49 50
cype@cype.com

www.cype.com

Avda. Eusebio Sempere, 5 – 03003 Alicante (Spain) • Tel : +34 965 92 25 50 • Fax: +34 965 12 49 50 • www.cype.com

CYPE Ingenieros, S.A.

Edited and printed in Alicante (Spain)

Presentation 5

CYPE-Connect 7

1. Introduction 7

2. Description 7

2.1. File 7

2.2. General data 7

2.3. Configuration 7

2.4. Help 8

3. Project introduction example 8

3.1. Project creation 8

3.2. Initial data generation assistant 8

3.3. Node creation 10

3.4. Edit the geometry and loads 11

3.5. Edit the joint 16

Presentation

CYPE-Connect has been created to provide users with a tool to design section connections in accordance with a series of predefined types within the program. Users can modify the thickness and dimensions of the plates, type of bolts, welds… etc. All these modifications are checked by the program, which in turn provides users with a report detailing any conditions that have not been verified.

CYPE-Connect

1. Introduction

CYPE-Connect has been created to provide users with a tool to design bolted and welded connections of rolled and welded steel I sections and coplanar hollow structural section connections such as those usually found in flat trusses (circular hollow sections, rectangular hollow sections, square hollow sections and hollow sections composed of two channels welded in a box), in accordance with a series of predefined types within the program. Users can modify the thickness and dimensions of the plates, type of bolts, welds, etc. All these modifications are checked by the program, which in turn provides users with a report detailing any conditions that have not been verified.

2. Description

The program consists of a general window which displays the joint of the selected node, if it has been solved. Along the top of the window are four drop-down menus:

2.1. File

This menu contains all the file management options (**New, File manager**, **Save**, **Save as** and **Job description**), Print (Job report and job drawings) and the Electronic license configuration.

2.2. General data

Within this menu are, first of all, the options to select and create the combinations and loadcases, together with their general design options. The next group of options contains the options to move from one joint to another. The two following options allow users to create the geometric description of the connection bars, how they are to be analysed, followed by their modification and verification.

Finally the **Check** and **Design** options are provided for the edited joint or for all the joints of the project.

2.3. Configuration

Located in the **Configuration** menu are the options to modify the design code for the joints, the units system to be applied, the configuration options for the size and orientation of the document, the printer selection and the styles to be used for the job reports.

Options are available to send the job, within which users can add contact addresses so they will appear when sending a job.

Using the **Drawings** option, users can create drawing formats as well as configure the peripherals where these formats will be used.

The **Details** option allows users to create a library by incorporating CAD files in DXF or DWG format, so they may be incorporated in the drawings.

Finally, an option is available to modify the background of the program working area.

2.4. Help

Contains all the help documents available for the program.

3. Project introduction example

The **About** option provides users with information on the program version and the license configuration being used.

3.1. Project creation

As with any program, use the **New** option from the **File** menu to begin creating a project. Introduce its name and description.

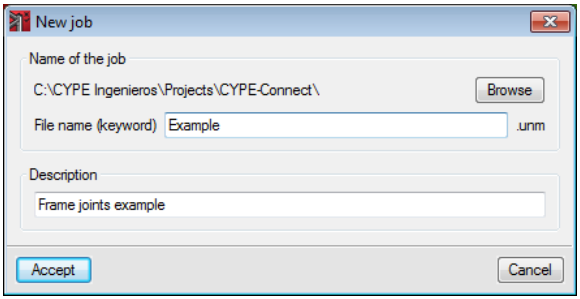

Fig. 1

3.2. Initial data generation assistant

Once the window has been accepted, the **New job** assistant is activated to complete the project data. The design code to be applied is selected in the first window that appears. In this case select Eurocodes 3 and 4.

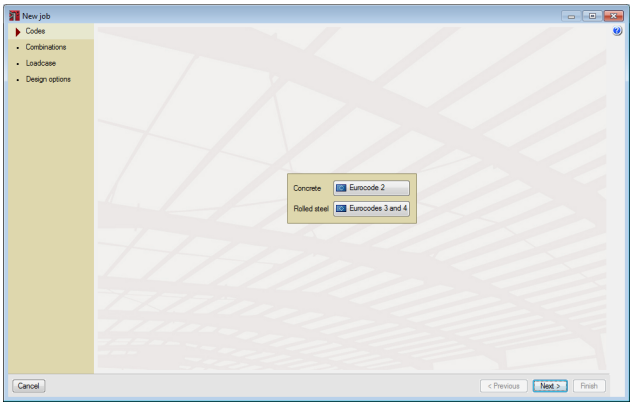

Fig. 2

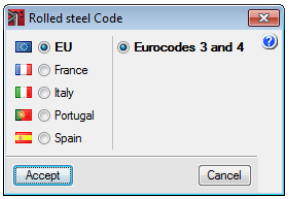

Fig. 3

The combination groups to solve the joint are defined.

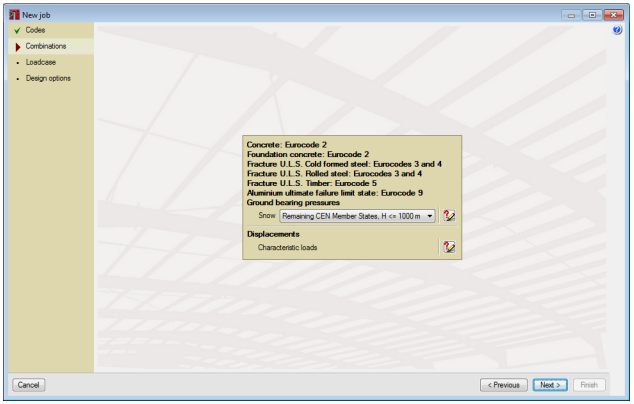

Fig. 4

Then the loadcases to be considered for the design and check of the joint are defined. For this example, there is one Self weight loadcase, one Live load loadcase and 6 Wind loadcases. To create them, click on the edit button of the type of loadcase to add.

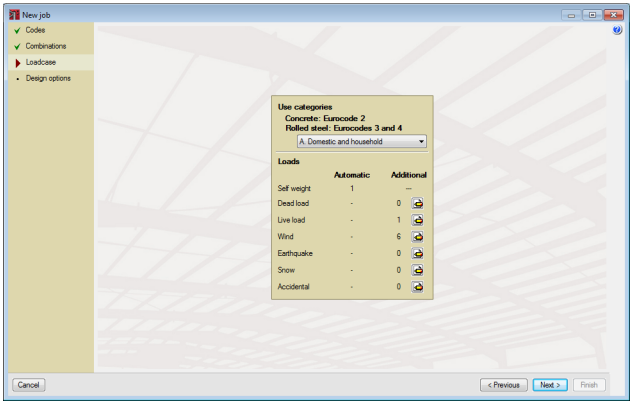

Fig. 5

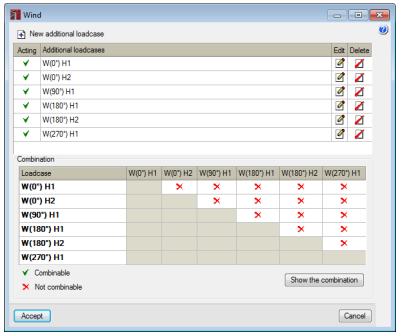

Fig. 6

Finally, select the connection method and the connection element configuration (bolts and stiffeners)

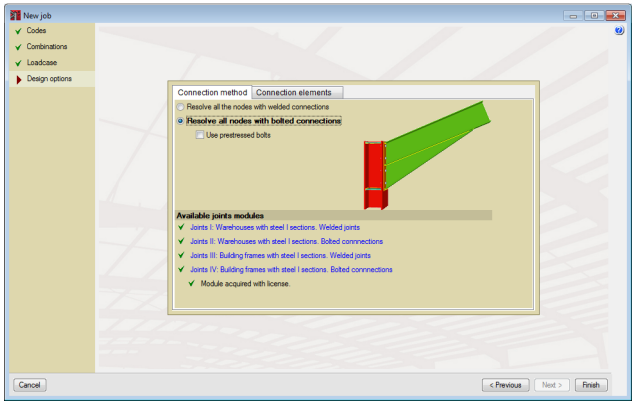

Fig. 7

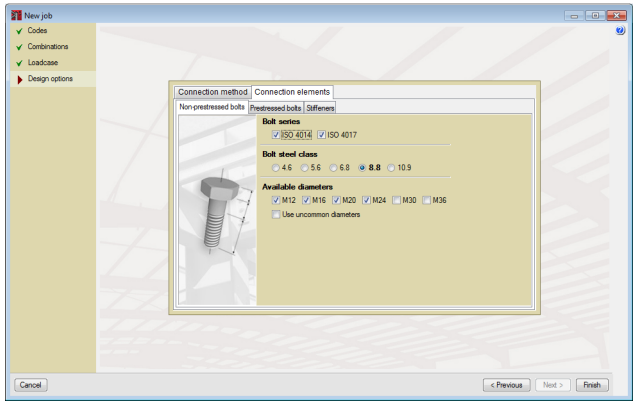

Fig. 8

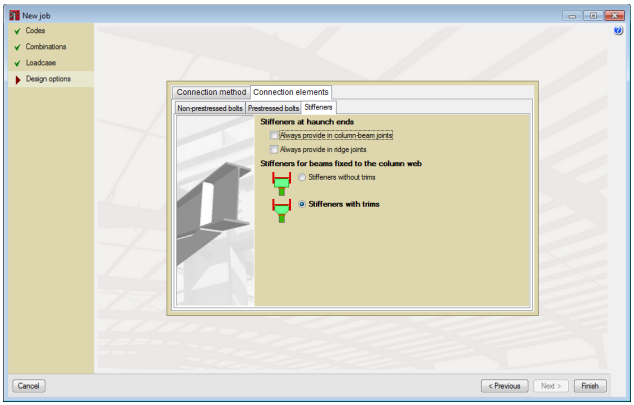

Fig. 9

3.3. Node creation

Once all the windows of the assistant have been accepted, the main window of the program appears. To define a new node type, click on the List of nodes option in the **General data** menu.

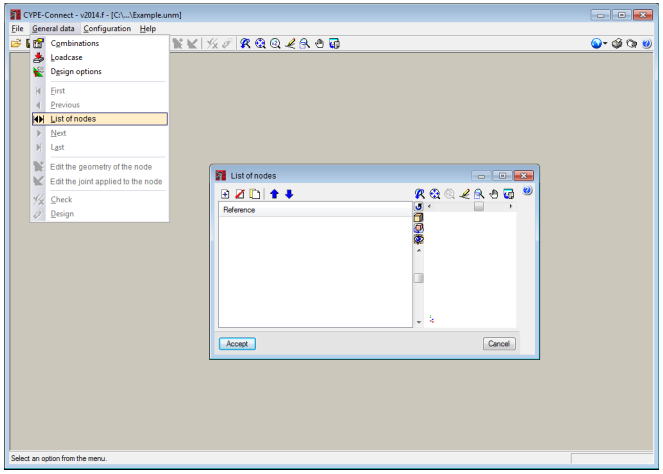

Fig. 10

By pressing on the  button, a new node is added. To help to introduce it, the program allows the node to be generated based on a series of predefined types, or alternatively, it can be introduced without any help. For this example, a predefined type will be used.

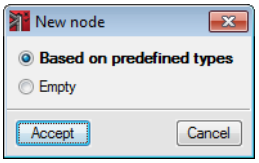

Fig. 11

The first joint to be defined will be the ridge node of the centre frame of the warehouse. To do so select **Splices**, then the ridge configuration and introduce an angle of 11.3° for both bar.

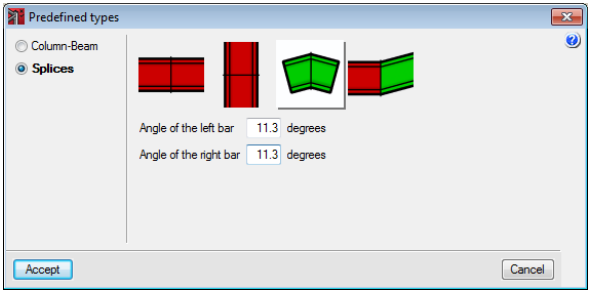

Fig. 12

Once the ridge node has been defined, the beam-column node is to be created. The same way as was done for the previous node, a new node will be defined, but in this case, indicate it is a **Column-Beam** node. Now select the type located on the first row, and in second place starting from the right.

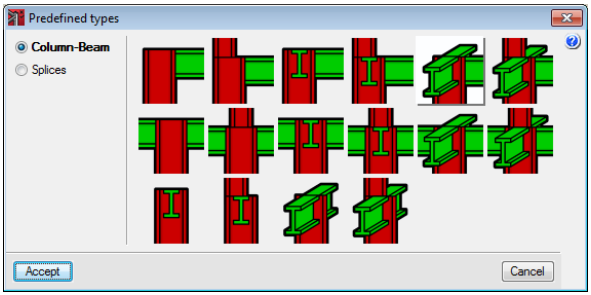

Fig. 13

3.4. Edit the geometry and loads

3.4.1. Node 1 (Ridge splice connection)

Once the window has been accepted, the next step consists in completing the definition of the two nodes, by indicating the series and section of each bar, as well as the loads which are to be considered in their design. To do so, click on the **Edit the geometry of the node** option in the **General data** menu.

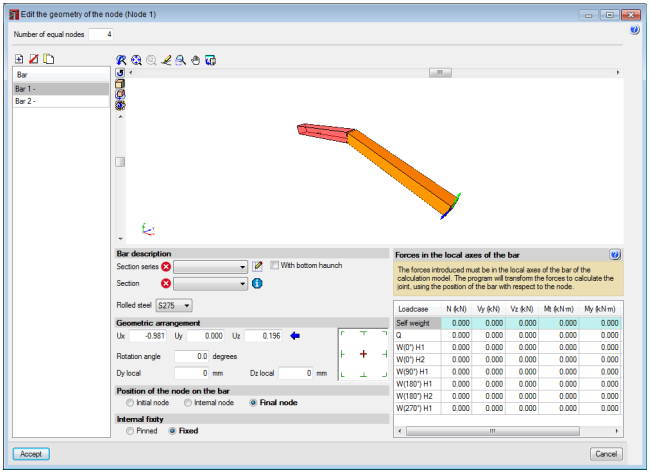

Fig. 14

Begin by indicating the number of equal node in the top left field of the window; in this case there are 4 equal ridge splice connections. The description of the connecting bars then has to be defined. The image on the right displays the bar being edited in a lighter tone as well as indicating its reference.

Now describe Bar 1, this is an IPE 300. To edit the section, select the series and section size in the **Bar description** section. As there is no default library, the sections to be used in this example have to be defined. Click on the **Edit the list of elements** icon . A new window called **Series of sections** opens which displays all the section series that are available for this project. To import the sections, click on the **Import of predefined section series** button , and a window will appear displaying all the manufacturer libraries the program can import. For this case, select IPE and HEB sections of ARCELOR.

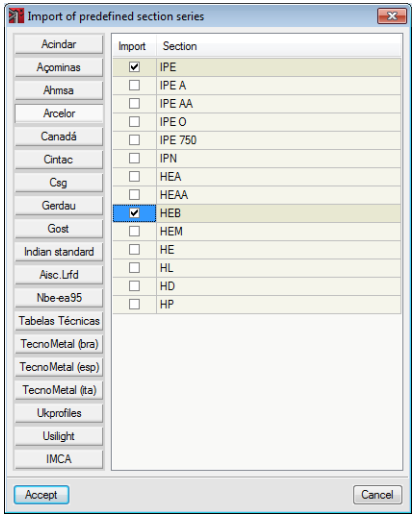

Fig. 15

Accept and select series IPE and section IPE 300 from the series. The next step to define this node consists in introducing the following loads:

Bar 1	N	Vy	Vz	Mt	My	Mz
Self weight	-6.263	0.000	1.253	0.00	-11.06	0.00
Q	-12.303	0.000	2.461	0.00	-21.72	0.00
W(0°) H1	10.203	0.000	-0.671	0.00	14.54	0.00
W(0°) H2	0.468	0.000	3.291	0.00	2.84	0.00
W(90°) H1	18.700	0.000	-3.740	0.00	9.27	0.00
W(180°) H1	9.677	0.000	-3.305	0.00	14.54	0.00
W(180°) H2	-0.834	0.000	-3.218	0.00	2.84	0.00
W(270°) H1	20.473	0.000	-4.095	0.00	11.41	0.00

Select Bar 2, define it in the same way as was done for Bar 1 and introduce the following loads:a

Bar 2	N	Vy	Vz	Mt	My	Mz
Self weight	-6.263	0.000	1.253	0.00	-11.06	0.00
Q	-12.303	0.000	2.461	0.00	-21.72	0.00
W(0°) H1	9.677	0.000	-3.305	0.00	14.54	0.00
W(0°) H2	-0.834	0.000	-3.218	0.00	2.84	0.00
W(90°) H1	18.700	0.000	-3.740	0.00	9.27	0.00
W(180°) H1	10.203	0.000	-0.671	0.00	14.54	0.00
W(180°) H2	0.468	0.000	3.291	0.00	2.84	0.00
W(270°) H1	20.473	0.000	-4.095	0.00	11.41	0.00

Accept and the geometry and loads of the nodes will be defined.

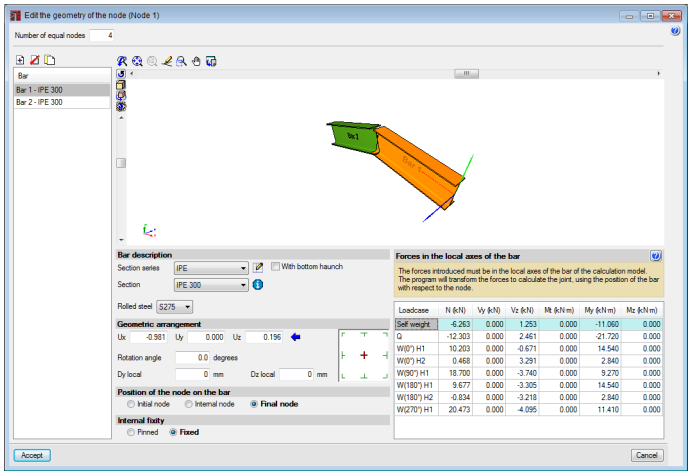

Fig. 16

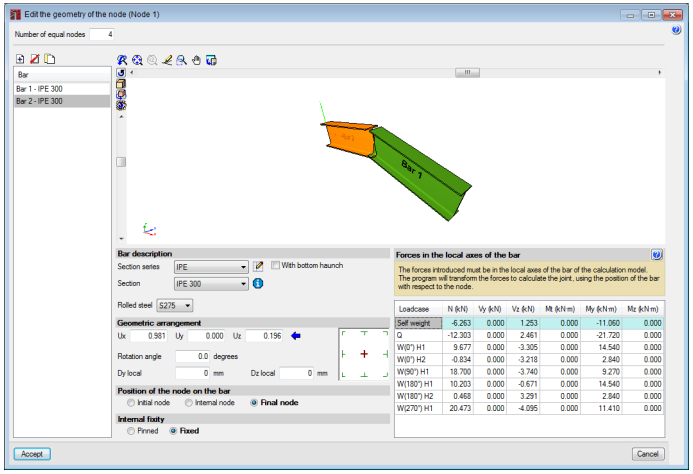

Fig. 17

3.4.2. Node 2 (Column-Beams)

Select Node 2, by pressing the right pointing black arrow on the top toolbar , and use the **Edit the geometry of the node** option from the **General data** menu again.

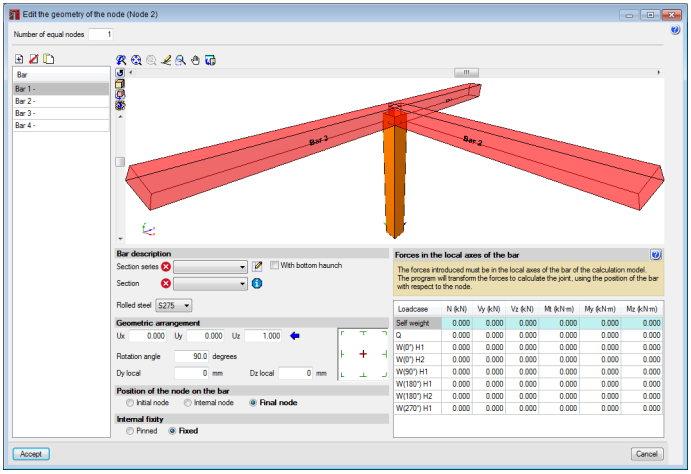

Fig. 18

Begin by describing Bar 1, which corresponds to the column of the joint. This will have an HEB 300 section. To edit the section, select the series and section in the **Bar description** section.

Having chosen the section and series, leave the remaining values as they are and proceed to select Bar 2, the beam that connects to the flange of the column. Similarly, define the beam as an IPE 300, and in the **Geometric arrangement** section, indicate the angle of the bar. This can be done in two ways, either by indicating the direction vector of the bar or indicate the angles that define the direction by clicking on the blue arrow . Click on the blue arrow and in this case the Elevation angle with respect to the horizontal plane will be 11.31°.

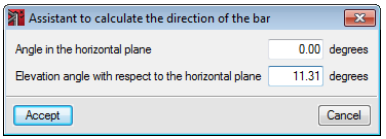

Fig. 19

Once the section and its direction have been defined, introduce the following loads for Bar 2:

Bar 2	N	Vy	Vz	Mt	My	Mz
Self weight	-8.269	0.000	-8.930	0.00	26.73	0.00
Q	-16.243	0.000	-17.539	0.00	52.51	0.00
W(0°) H1	10.203	0.000	15.298	0.00	-44.08	0.00
W(0°) H2	0.468	0.000	-0.588	0.00	-11.03	0.00
W(90°) H1	18.700	0.000	13.144	0.00	-36.69	0.00
W(180°) H1	9.677	0.000	11.769	0.00	-28.92	0.00
W(180°) H2	-0.834	0.000	3.587	0.00	1.50	0.00
W(270°) H1	20.473	0.000	15.094	0.00	-42.39	0.00

Select Bar 3 and similarly, indicate it is an IPE 100. Within the **Internal fixity** section, select **Pinned**, and then define the following loads:

Bar 3	N	Vy	Vz
Self weight	0.495	0.000	-0.159
Q	0.919	0.000	0.000
W(0°) H1	20.626	0.000	0.000
W(0°) H2	20.718	0.000	0.000
W(90°) H1	-4.141	0.000	0.000
W(180°) H1	18.994	0.000	0.000
W(180°) H2	19.961	0.000	0.000
W(270°) H1	-5.578	0.000	0.000

Last of all; define Bar 4 with an IPE 100 section. Select Pinned within the **Internal fixity** section. For this last case, select **Final node** in the **Position of the node on the bar** section, and change the sign of the Uy component of the direction vector of the bar so it coincides with Metal 3D from which we have obtained the forces.

Bar 4	N	Vy	Vz
Self weight	0.493	0.000	0.159
Q	0.915	0.000	0.000
W(0°) H1	20.537	0.000	0.000
W(0°) H2	20.628	0.000	0.000
W(90°) H1	-4.841	0.000	0.000
W(180°) H1	18.915	0.000	0.000
W(180°) H2	19.878	0.000	0.000
W(270°) H1	-4.837	0.000	0.000

Having defined the geometry and loads, the node is completely defined to be able to apply the joint.

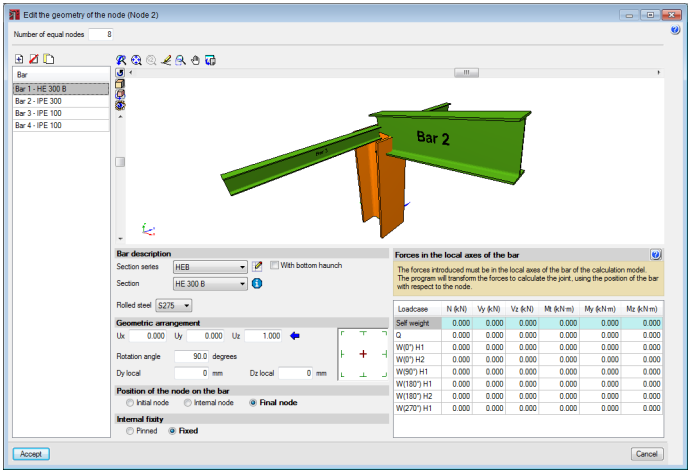

Fig. 20

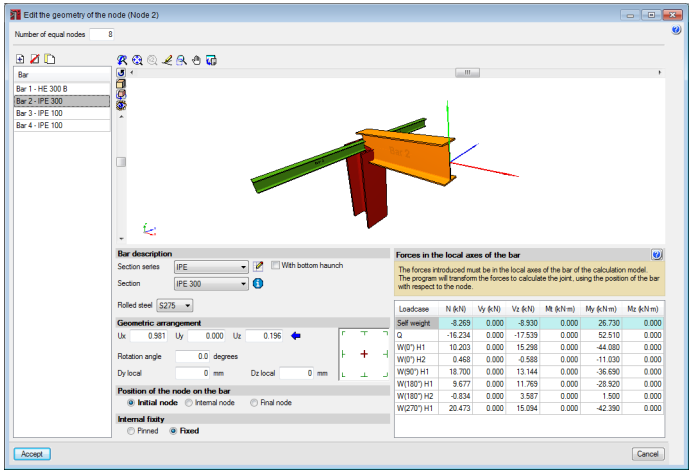

Fig. 21

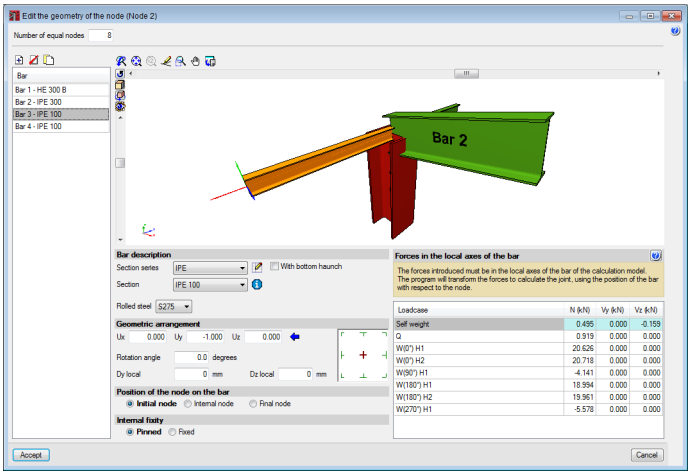

Fig. 22

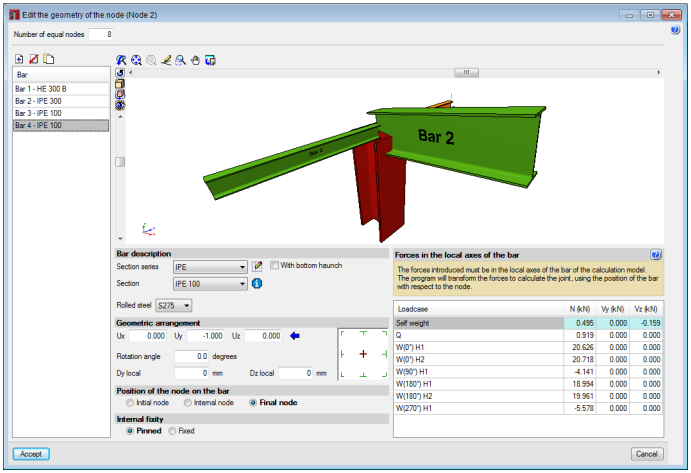

Fig. 23

Having defined the node data, click on **Design** in the **General data** menu and indicate that all the elements are to be designed.

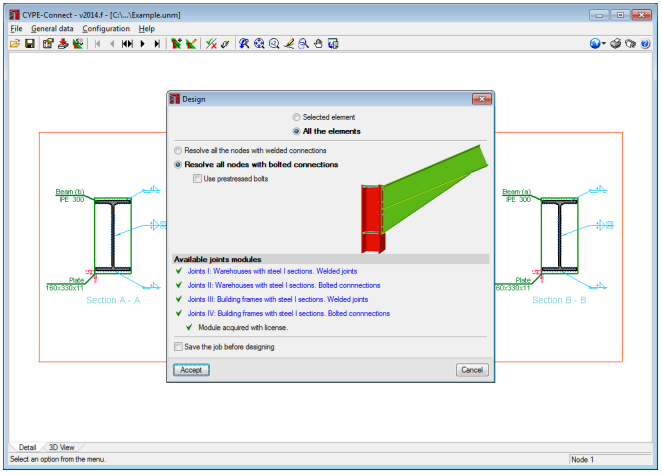

Fig. 24

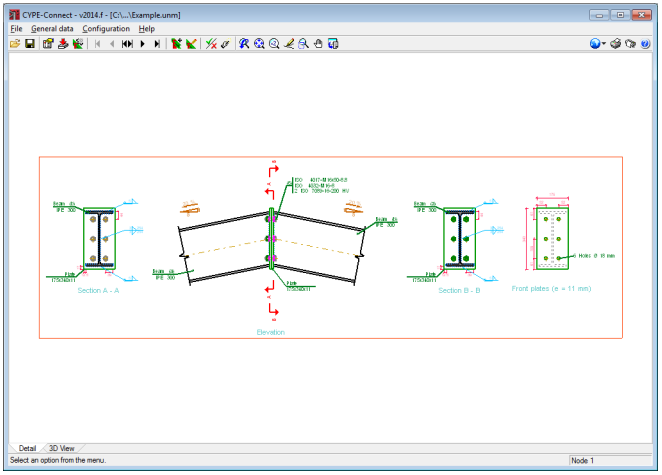

Fig. 25

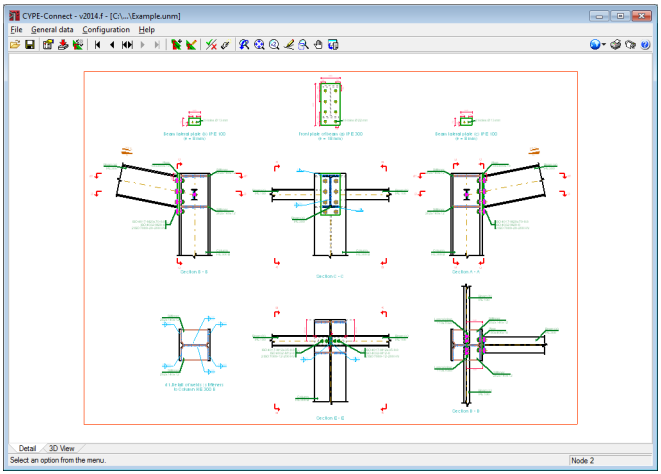

Fig. 26

When Node 2 is edited, the joint editor window is displayed. On the left of the window is a list of the components of the joint together with their individual edit buttons. To the right of the list is the graphical 3D view of all the modifications that have been carried out on the mode. Below the image is where any errors that have been encountered upon designing the node are displayed. Lastly, along the top are the **Design, Code check, Complete report of the node** and **Detailing buttons**.

Now modify the joint, to leave the IPE-300-HEB300 as a fixed connection and the IPE 100s as pinned connections. To do so, edit the IPE 300 Beam (a) and deactivate the bolted connection option.

3.5. Edit the joint

Here users can see the connections of the joint that the program has designed based on the data that has been introduced. They can be modified using the **Edit the joint applied to the node** option in the **General data** menu.

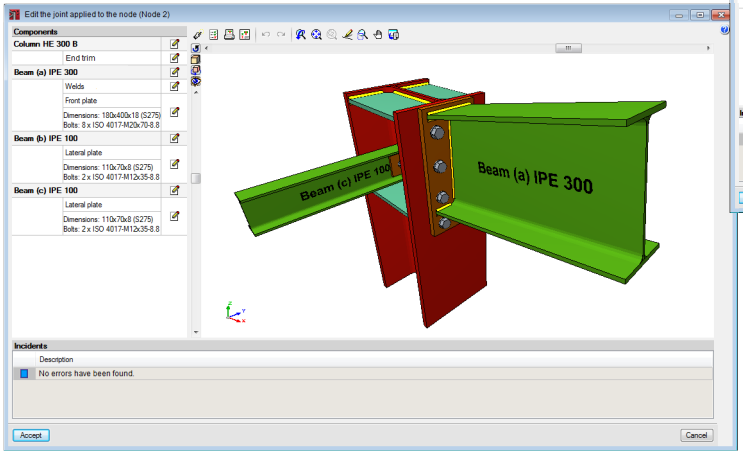

Fig. 27

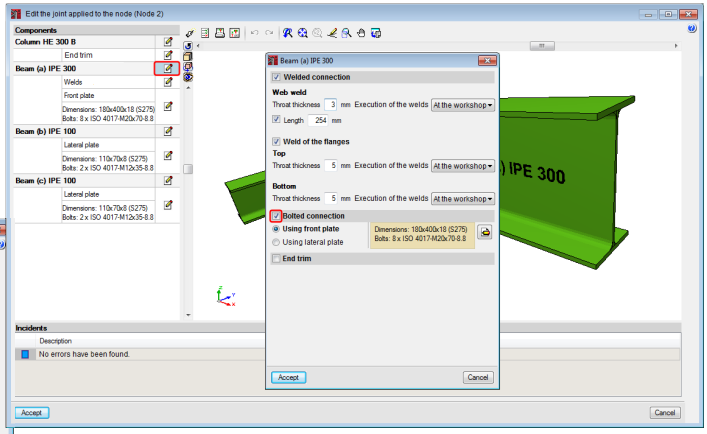

Fig. 28

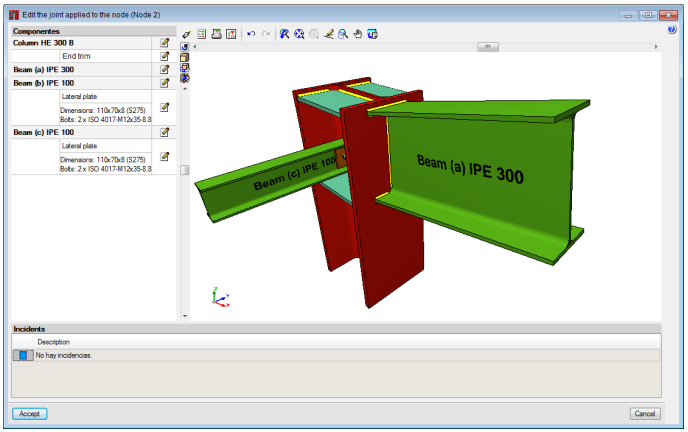

Fig. 29

Use the **End trim** option indicating the trim is to be carried out in accordance with the IPE 300 column. The column is then trimmed along a plane parallel to the flange of the beam.

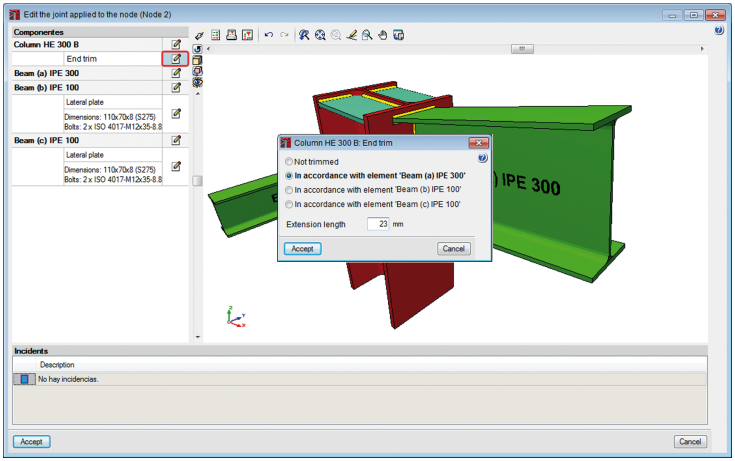

Fig. 30

Now the stiffeners will have to be adjusted to the new geometry of the HEB 300 column. To do so edit the HEB 300 column.

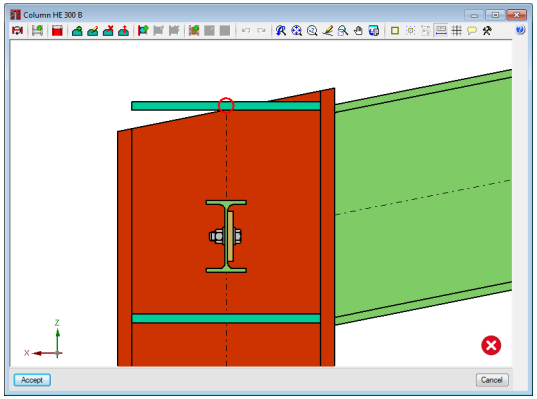

Fig. 31

The following options to modify the joint can be seen along the top of the window:

Reverse the viewpoint. Using this option, the elements located on the opposite side of the beam can be seen and modified.

Generate reinforcement elements. If there are no reinforcement elements, the program automatically generates them.

End trim. This option opens the window we saw previously, to indicate how the column ends.

Introduce stiffener. This can be done in two ways. The first consists in selecting the blue dot located at the intersection of the beam and the internal side of the column flange and moving the cursor to the opposite flange. The program will introduce the stiffener. The second method consists in selecting the yellow dot that is surrounded by a red circle located at the intersection of the beam flange and column

flange. If the cursor is placed near the dot, the introduction possibilities will be displayed. If it is clicked on, the yellow dot on the opposite flange will then have to be selected to define the position of the other end of the stiffener.

Edit stiffeners. Allows for several stiffeners to be edited at once to modify their dimensions and welds.

Delete stiffeners.

Move stiffeners.

Introduce new reinforcement element for moment connection at the web. Introduces a vertical reinforcement element between the horizontal stiffeners to aid in the connection of the beams with the web of the column.

Edit reinforcement element for moment connection at the web. By selecting the connection plate between the beam and stiffener, a new window opens in which the dimensions, position and welds of the connection plate and reinforcement can be edited.

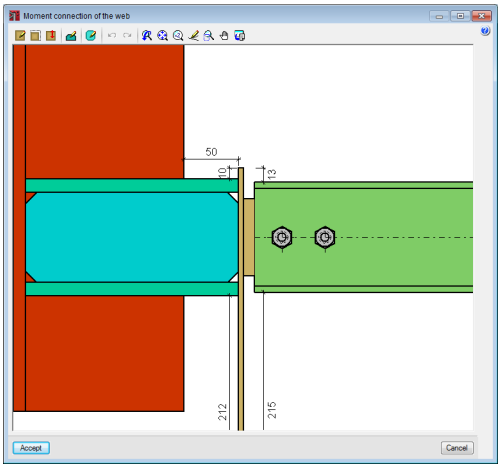

Fig. 32

Delete reinforcement element for moment connection at the web.

Introduce reinforcement plate at the web. To introduce a reinforcement plate, the two stiffeners must be selected, between which the web of the column will be reinforced.

Edit the reinforcement plate at the web. If the plate is selected, users can modify its thickness, material and welds.

Delete the reinforcement plate at the web.

To continue with the example, use the **Delete stiffeners** option to delete the stiffener. Then click on the option to reintroduce it and select the yellow point with the red circle.

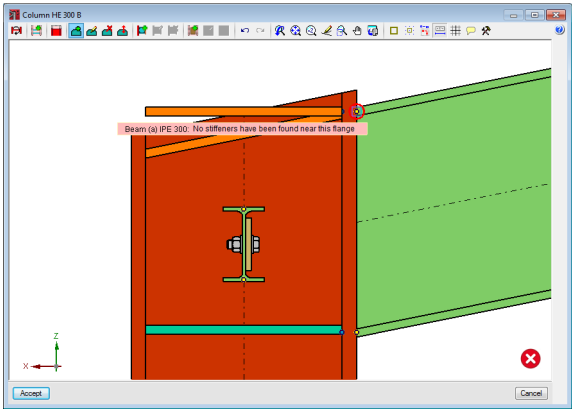

Fig. 33

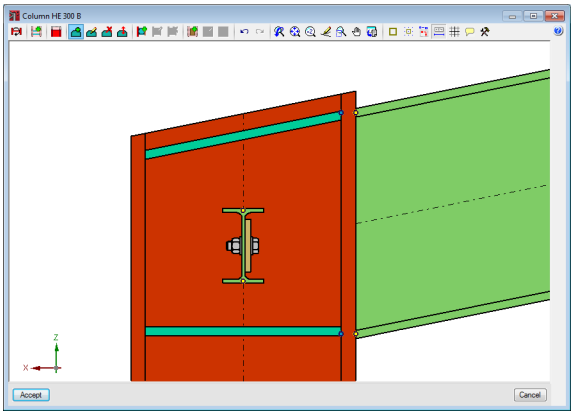

Fig. 34

Accept and the modification will be saved. Now click on the Code Check button to see if there are any errors.

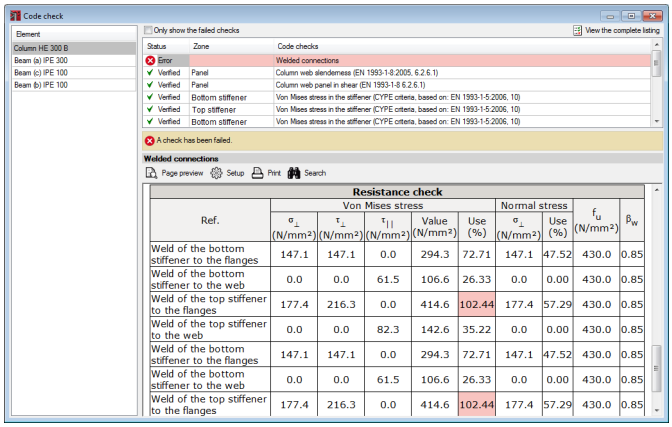

Fig. 35

The report indicates that the welds between the top stiffener and column flanges have not been verified. It also indicates the thicknesses of the welds are insufficient. To correct this, edit the HEB 300 column and using the option, select the stiffeners and modify the throat thickness of the weld to 6mm.

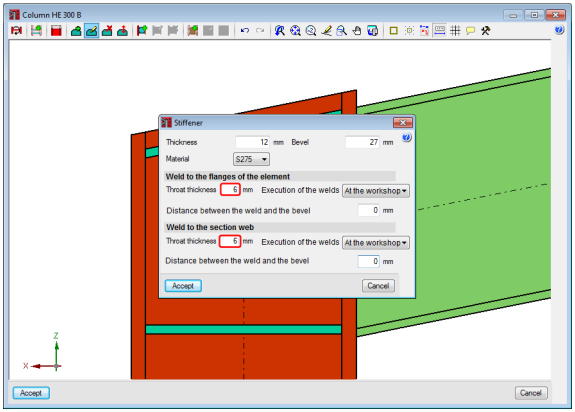

Fig. 36

Now run a code check; all the modifications that have been carried out on Node 2 are verified.

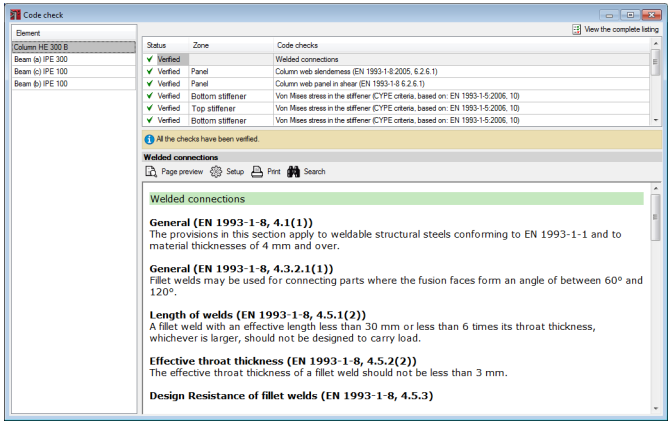

Fig. 37

To print the report, click on **File > Print > Job Report**. A new window appears indicating the elements to be displayed in the report.

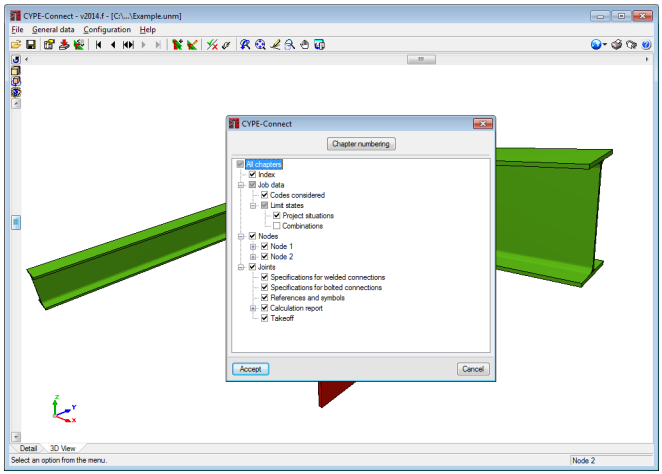

Fig. 38

Upon accepting, a preview window appears of the document to print.

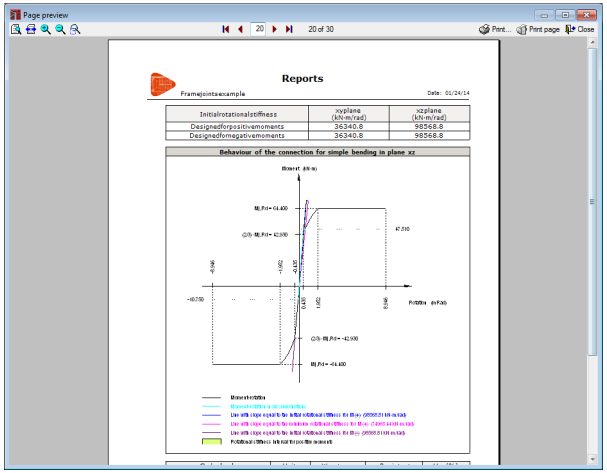

Fig. 39

To obtain the drawings of the job, click on **File > Print > Job Drawings**. The **Drawing selection** window will open. Add a drawing and select the scales to be applied to the elements in the list.

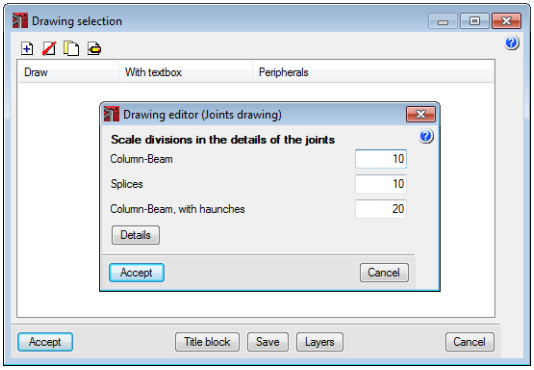

Fig. 40

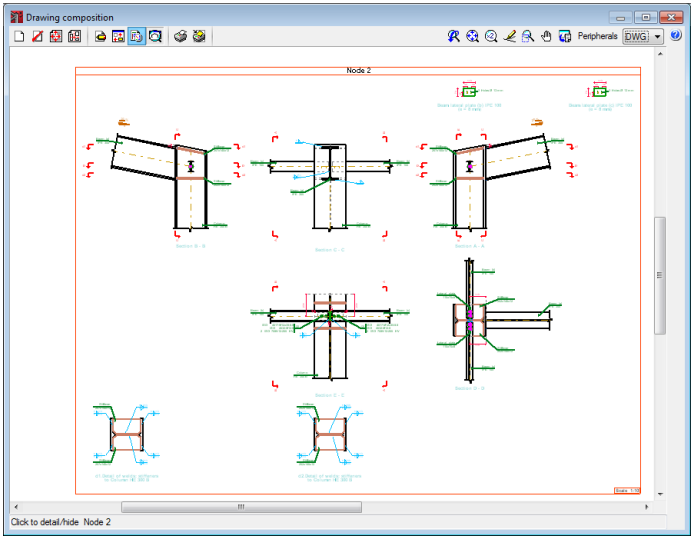

Fig. 41