BULK STORAGE SILO

CivilFEM

Introduction

Storage silos have been present all over the world for more than a century.

They are present in transport chains from the beginning of the chain, as well as in intermediate stages and obviously, at the end of the process. In all of these stages, silos are used as storage devices.

In this case study we are going to modelize the behaviour when an earthquake occurs.

Data

Bulk:

- Material: Concrete
- Density: 16 kN/m3

Silo:

- Diameter: 2,4 m
- Height: 8,0 m

Hopper:

- Diameter: 2,4 m
- Discharge hole: 0,20 m

Top:

- Height: 0,15 m

Materials:

- Steel S 275 JR
- Steel S 355 JR

Steel section:

- 2 UPN 120 (Pilars)
- Circular 8 mm (Braces)

Ground :

- Aceleration: 0,25·g
- Type B

CivilFEM powered by Marc

With CivilFEM powered by Marc we can perform a complete analysis of the silo.

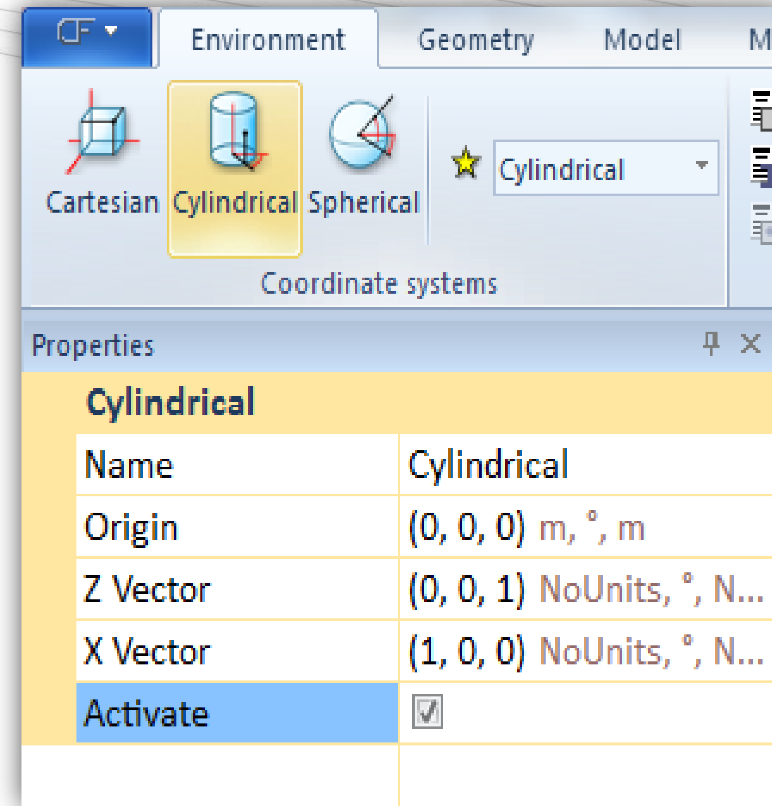
All design stages are performed with CivilFEM powered by Marc, from geometry definiton to the solving step.

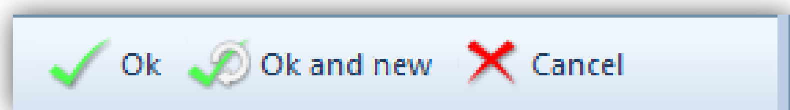
It is also possible to make a seismic analysis according to Eurocode 8 EN 1998–1: 2004.

Geometry

First of all, for convenience shake we should change the coordinate system from Global Cartesian to Cylindrical Coordinates and place the origin at (0,0,0).

1. Click on **Enviroment** tab.
2. Enter the parameter values.
3. Check the **Activate** box.
4. Click on **OK**.



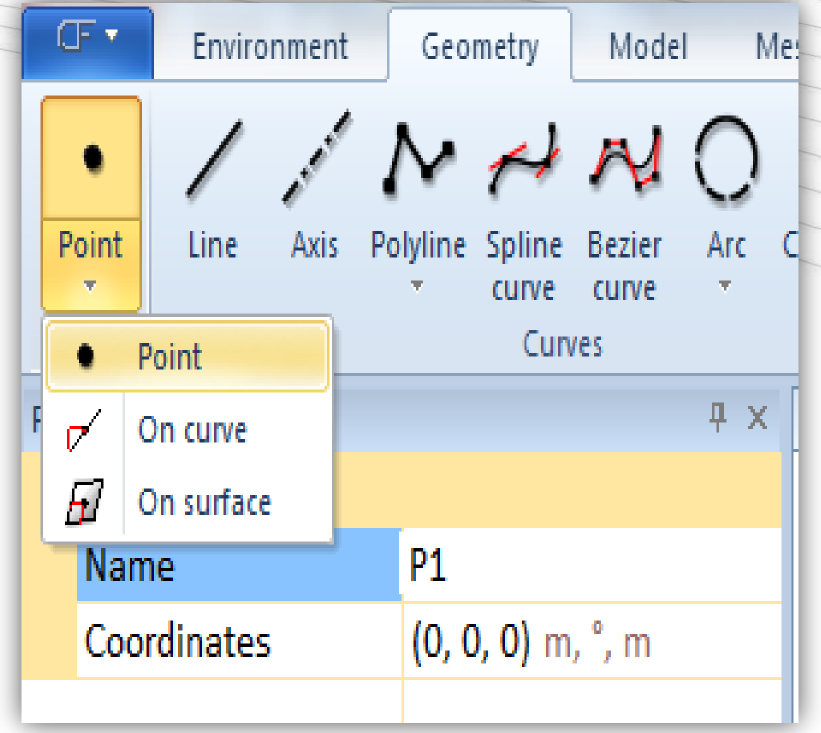


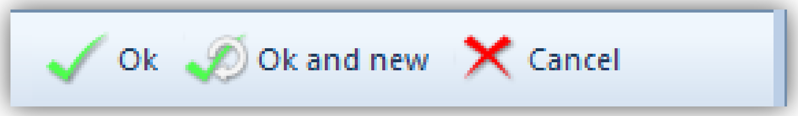
Points

In order to create the points, follow these steps:

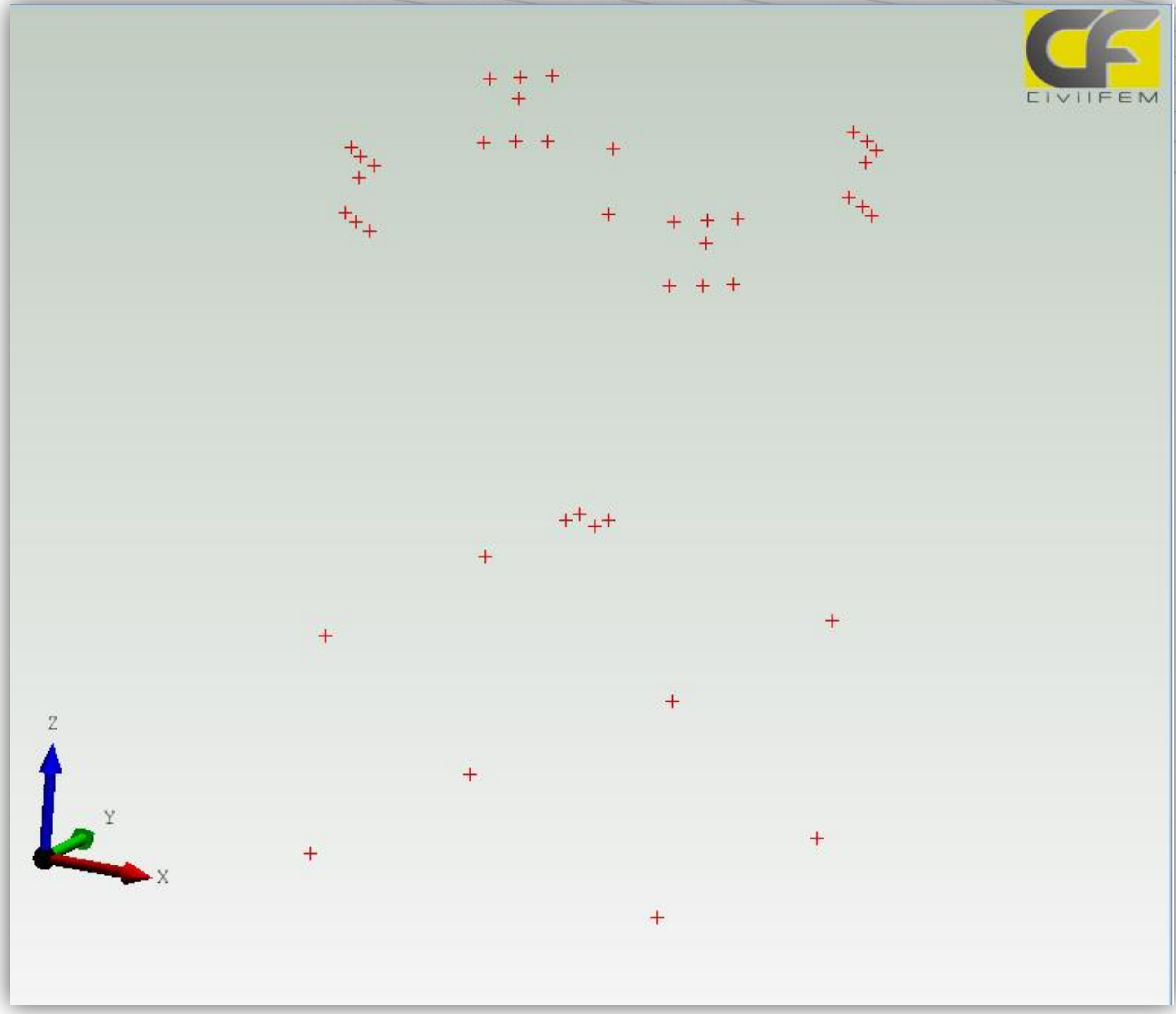
1. Click on **Geometry** tab.
2. Click on **Point** tab.
3. Click on **Point** again.
4. Enter the **name** and **coordinates**.
5. Click on **OK**.

Remember that we are in the Cylindrical system: **(x, θ, z)**





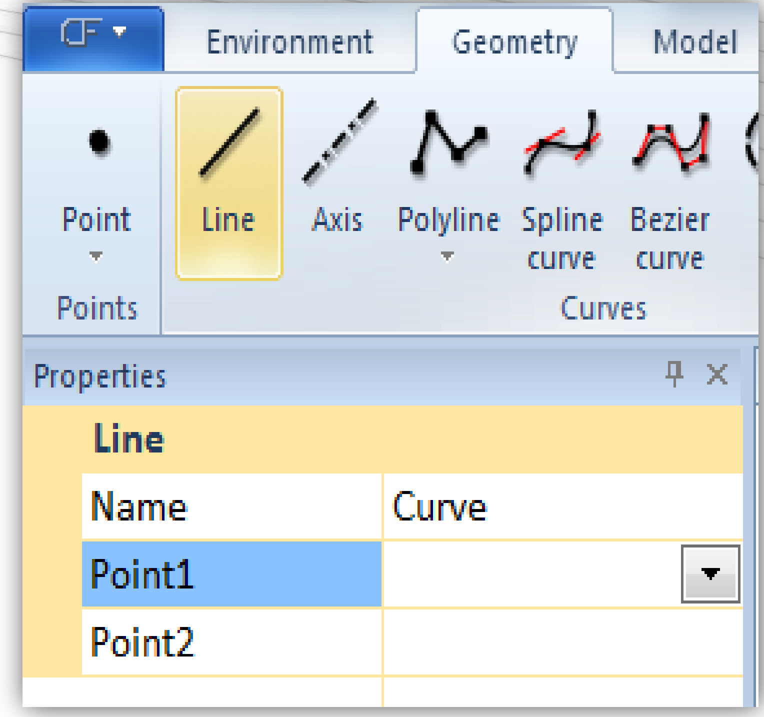
Points



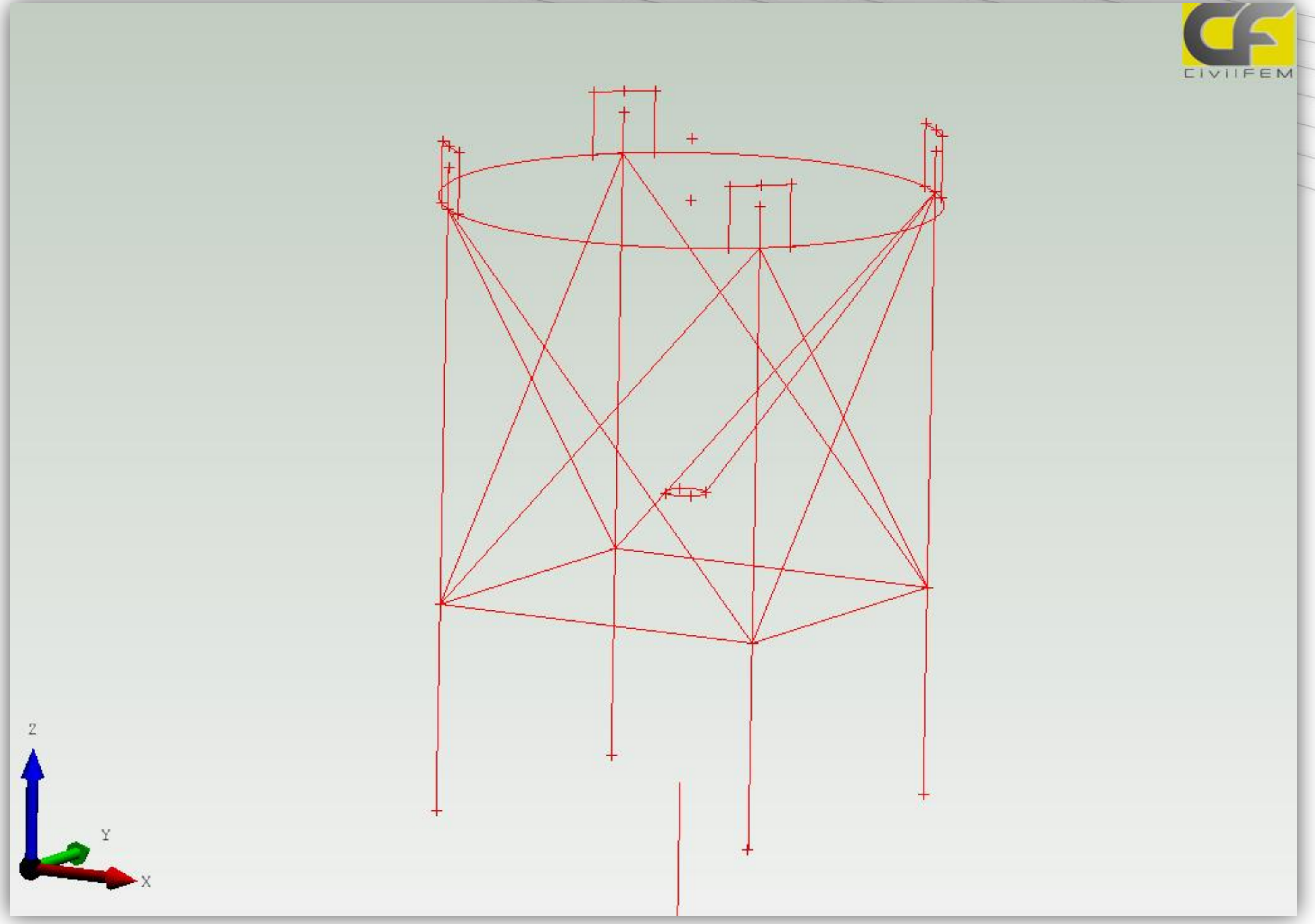
Lines

To create the lines:

1. Click on **Geometry** tab.
2. Click on **Line** tab.
3. Check the **Referenced** box.
4. Enter the line name.
5. Select the points that define the line.
6. Click on **OK**.



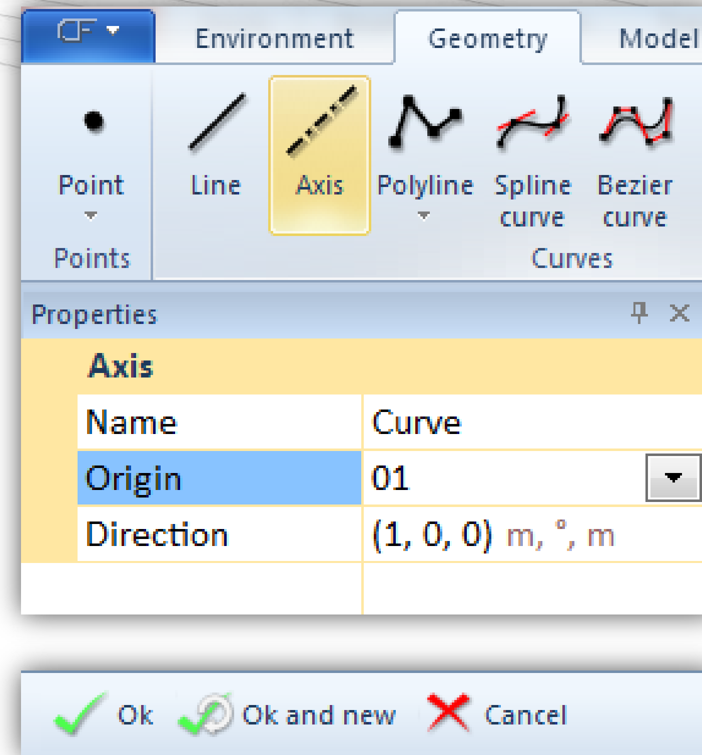
Lines



Axis

To create the Axis:

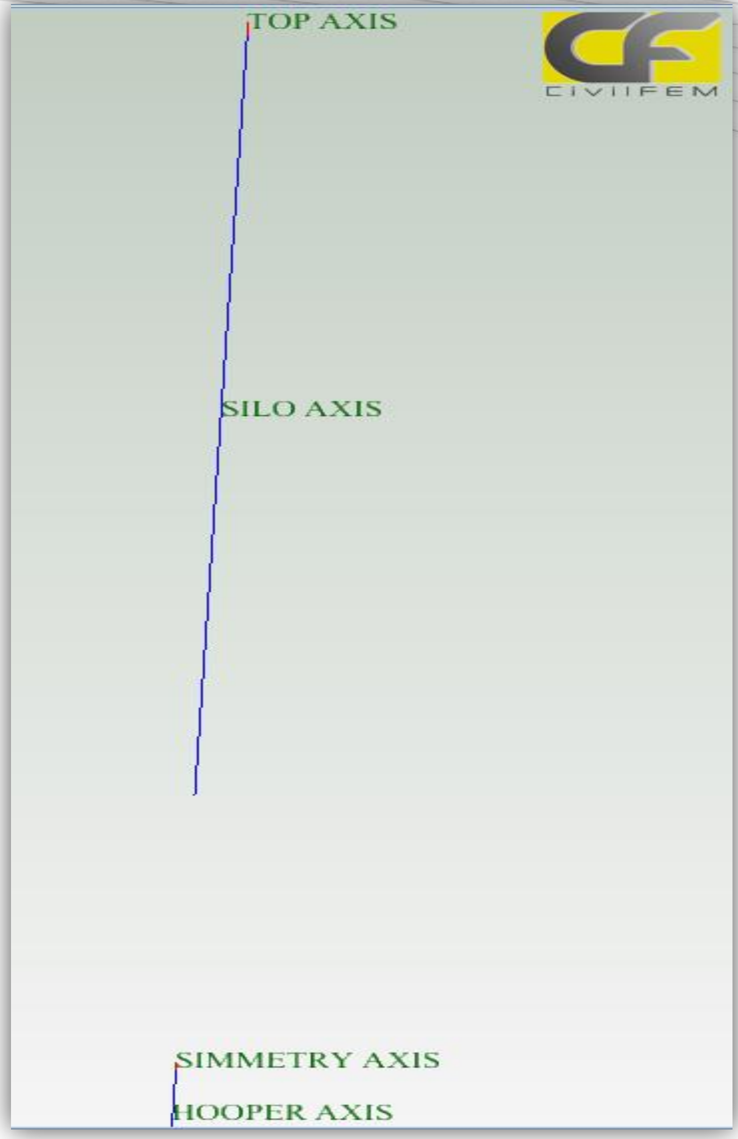
1. Click on **Geometry** tab.
2. Click on **Axis** tab.
3. Enter the axis **Name**.
4. Enter the coordinates of the **Direction Vector**
5. Click on **OK**.



Create four axis:

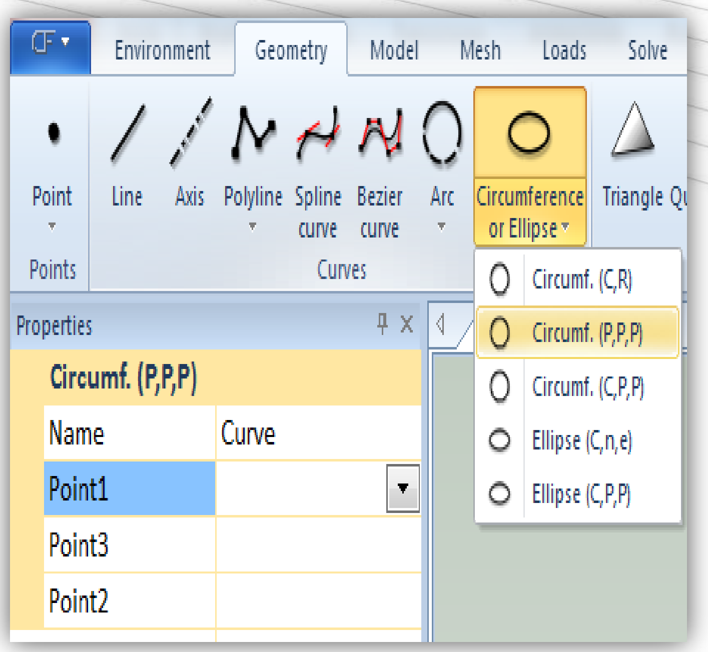
Silo body, hopper body, top body and a symmetry axis.

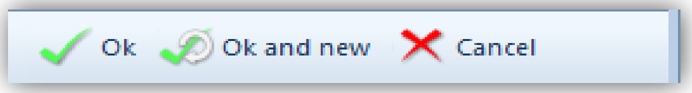
Axis



Circles

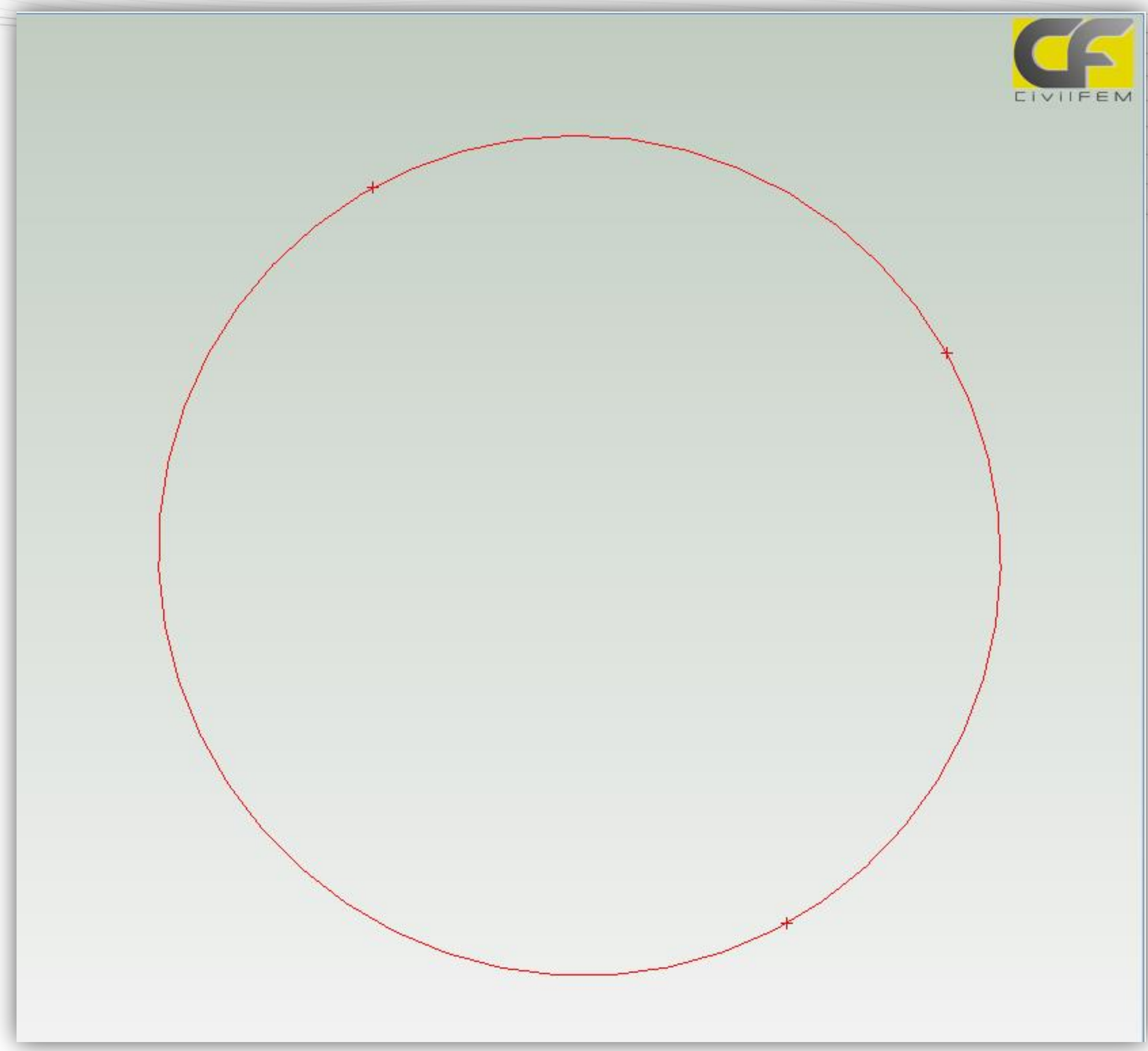
1. Click on **Geometry** tab.
2. Click on **Circumference or Elipse** tab.
3. Click on **Circum. (P, P, P)**.
4. Enter the axis **Name**.
5. Enter **Start vertex**, **End point** and **coordinates** of the **center**.
6. Click on **OK**.





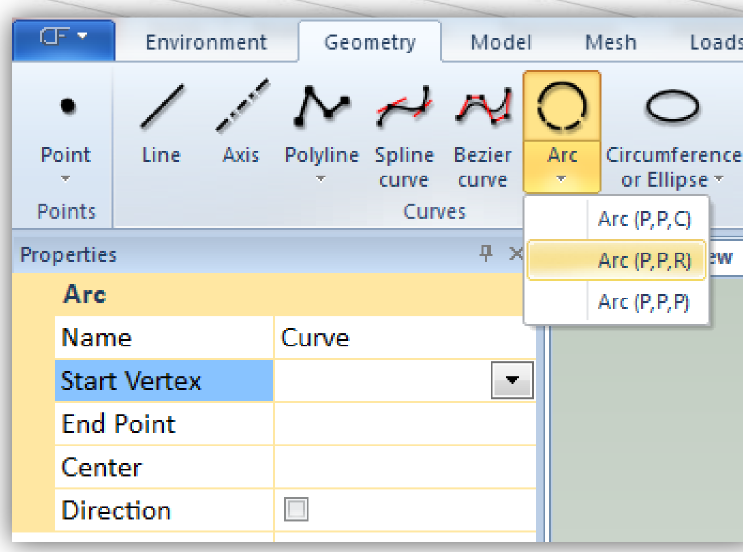
Check the **Direction** box.

Circles

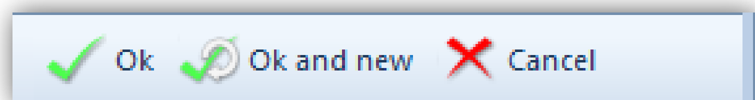


Arcs

1. Click on **Geometry** tab.
2. Click on **Arc** tab.
3. Enter the axis **Name**.
4. Enter **Start vertex**, **End point** and **coordinates** of the **center**.
5. Click on **OK**.



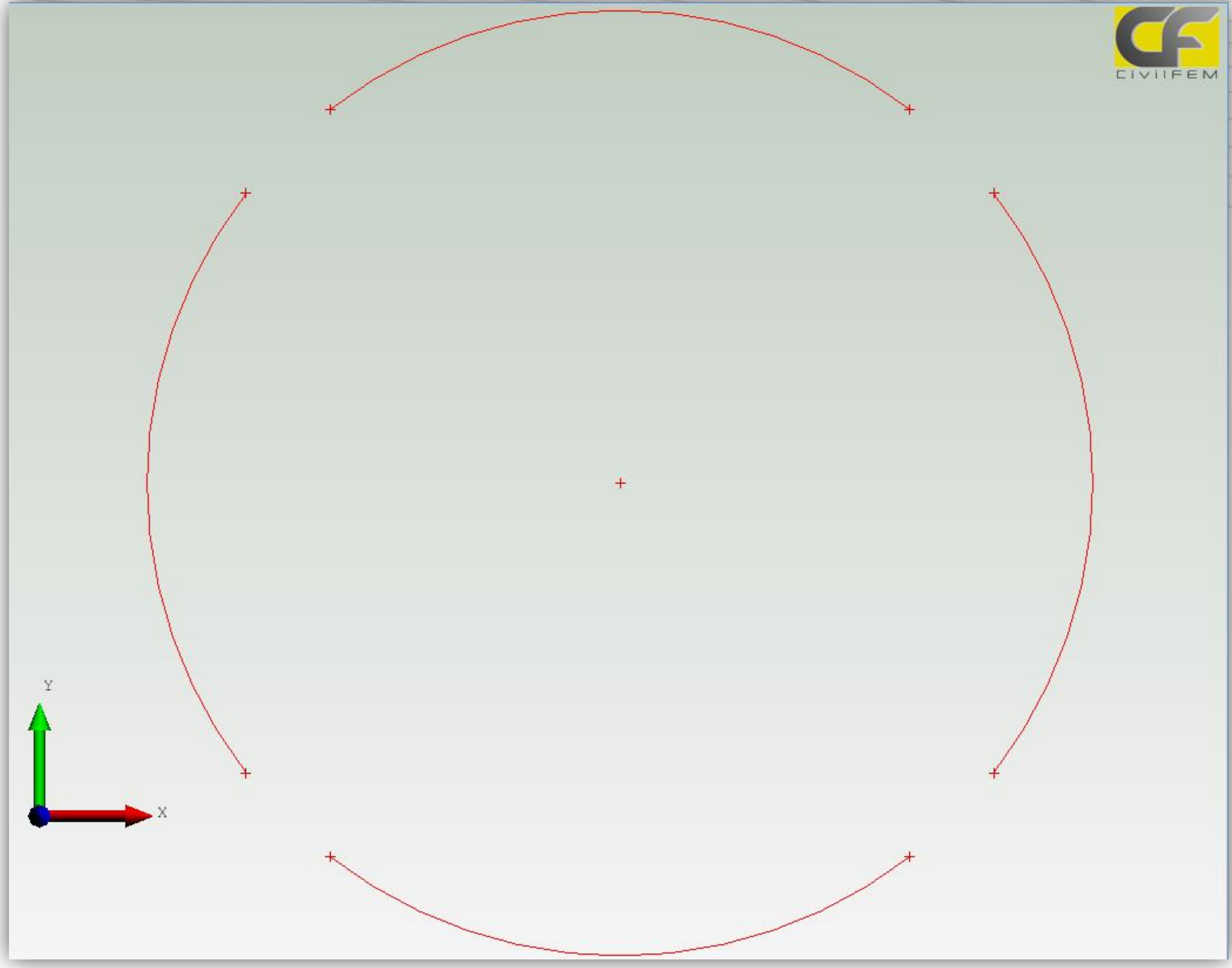
Check the **Direction** box.



INGECIBER, S.A.

CivilFEM

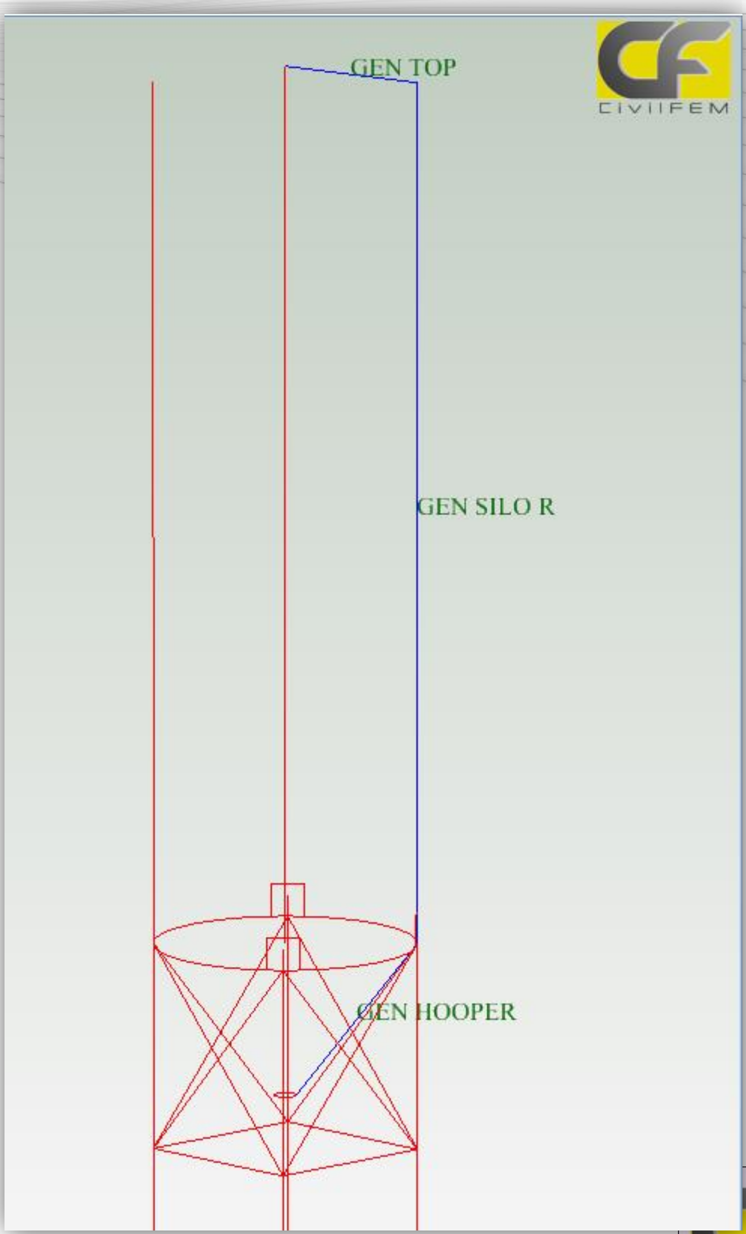
Arcs



Generatrix

Now create three generatrices for the revolution surfaces that will form the body of the silo, hopper and top body.

The procedure is the same as the procedure for defining lines.

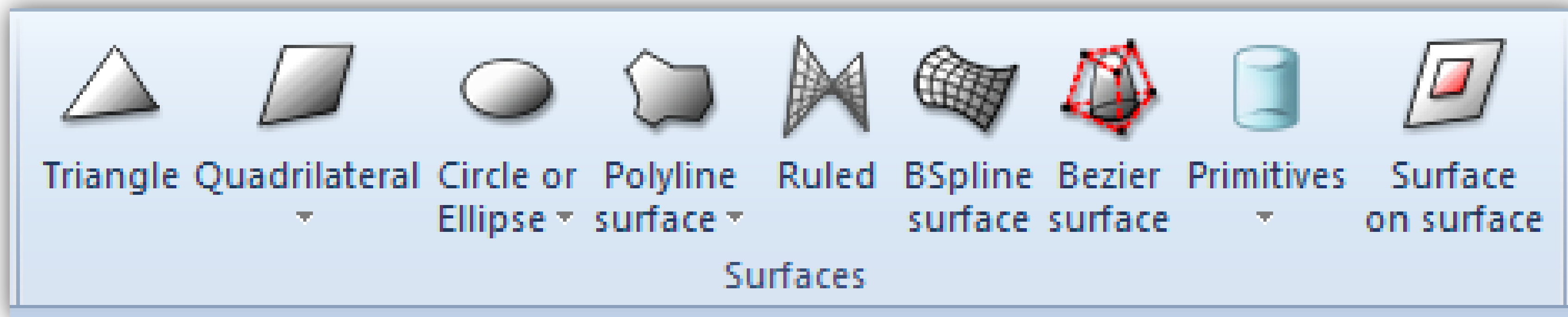


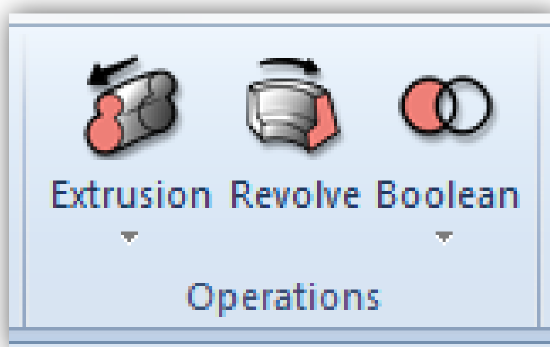
INGECIBER, S.A.

Surfaces

In CivilFEM with Marc we have different ways to create surfaces.

Clicking on the Geometry tab, different tools for the creation of surfaces are shown

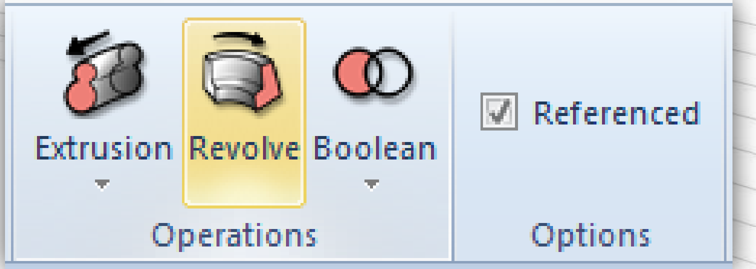


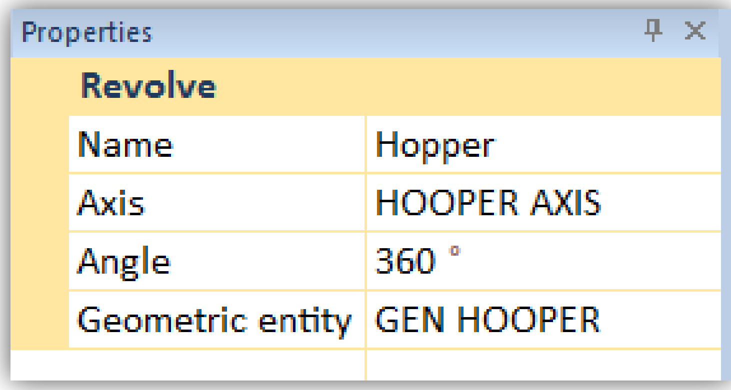


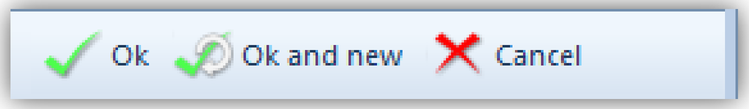
Surfaces

For defining the silo, hopper and top body, follow these steps:

1. Click on **Geometry** tab.
2. Click on **Revolve** tab.
3. Check the **Reference** box.
4. Enter **name**, **axis**, **angle** of revolution and the **generatrix**, in this order.
5. Click on **OK**.



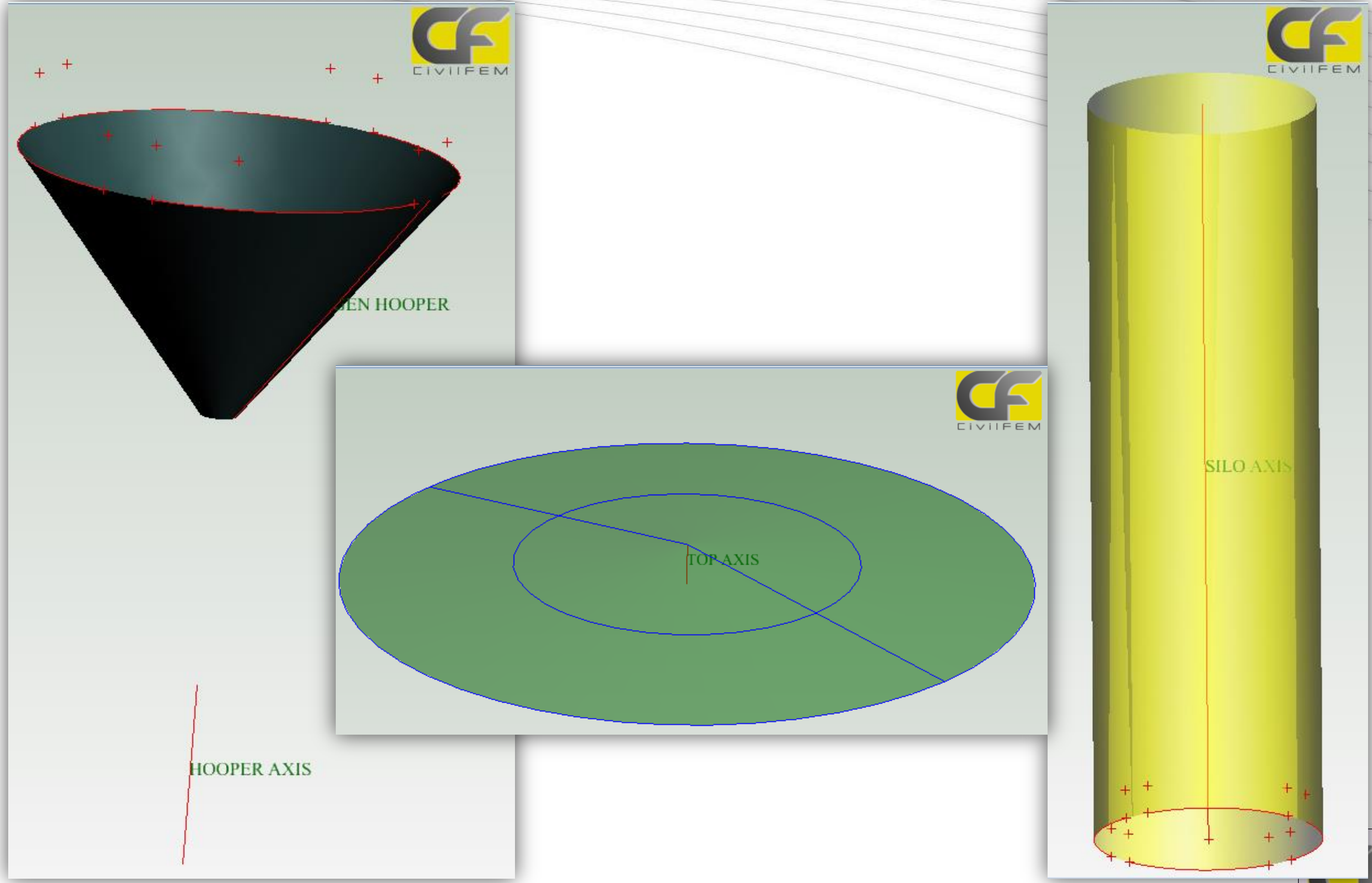




INGECIBER, S.A.

CivilFEM

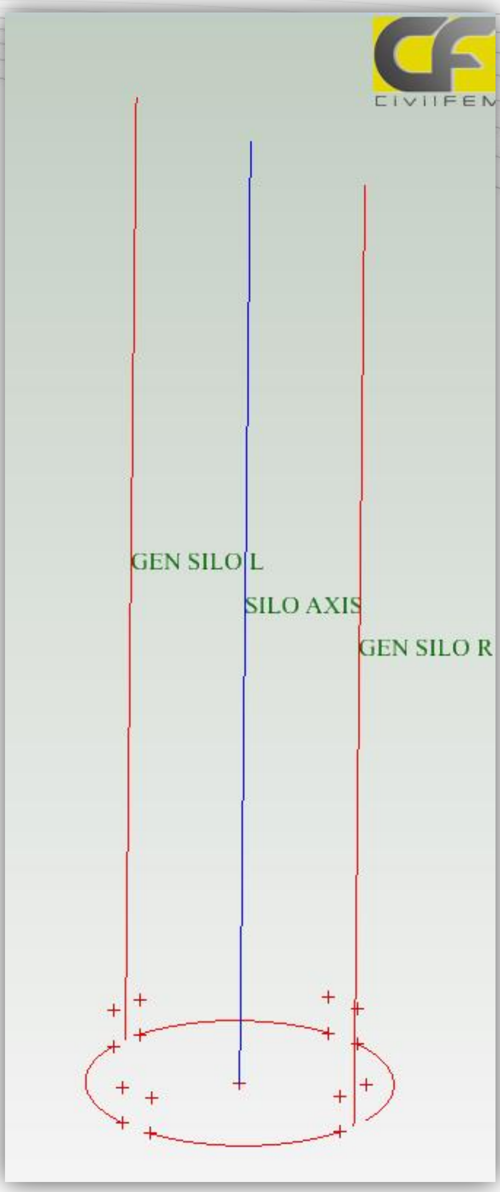
Surfaces



Surfaces

Futhermore, we need to create two auxiliary surfaces to apply the wind load.

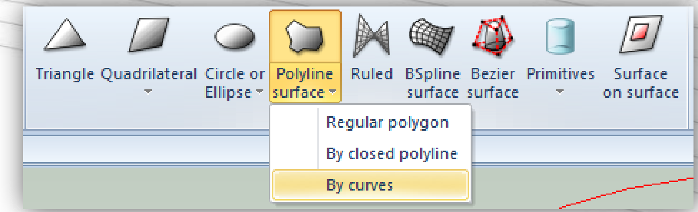
This surfaces will be defined by two half-cylinders created by revolution of two lines rotated 180º.

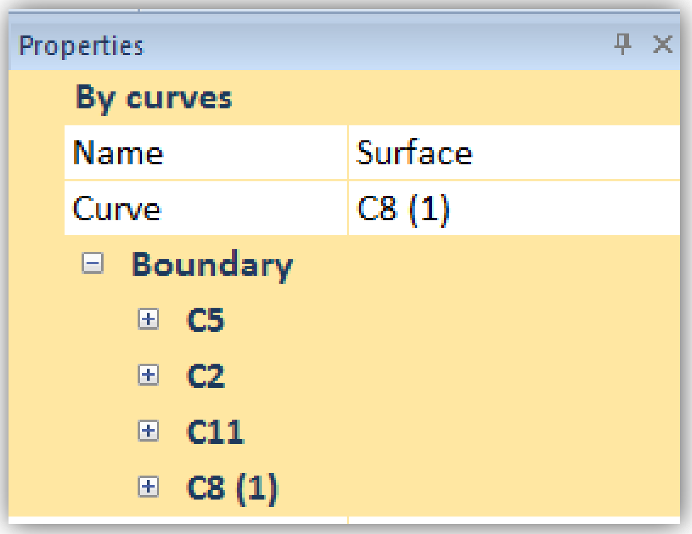


Surfaces

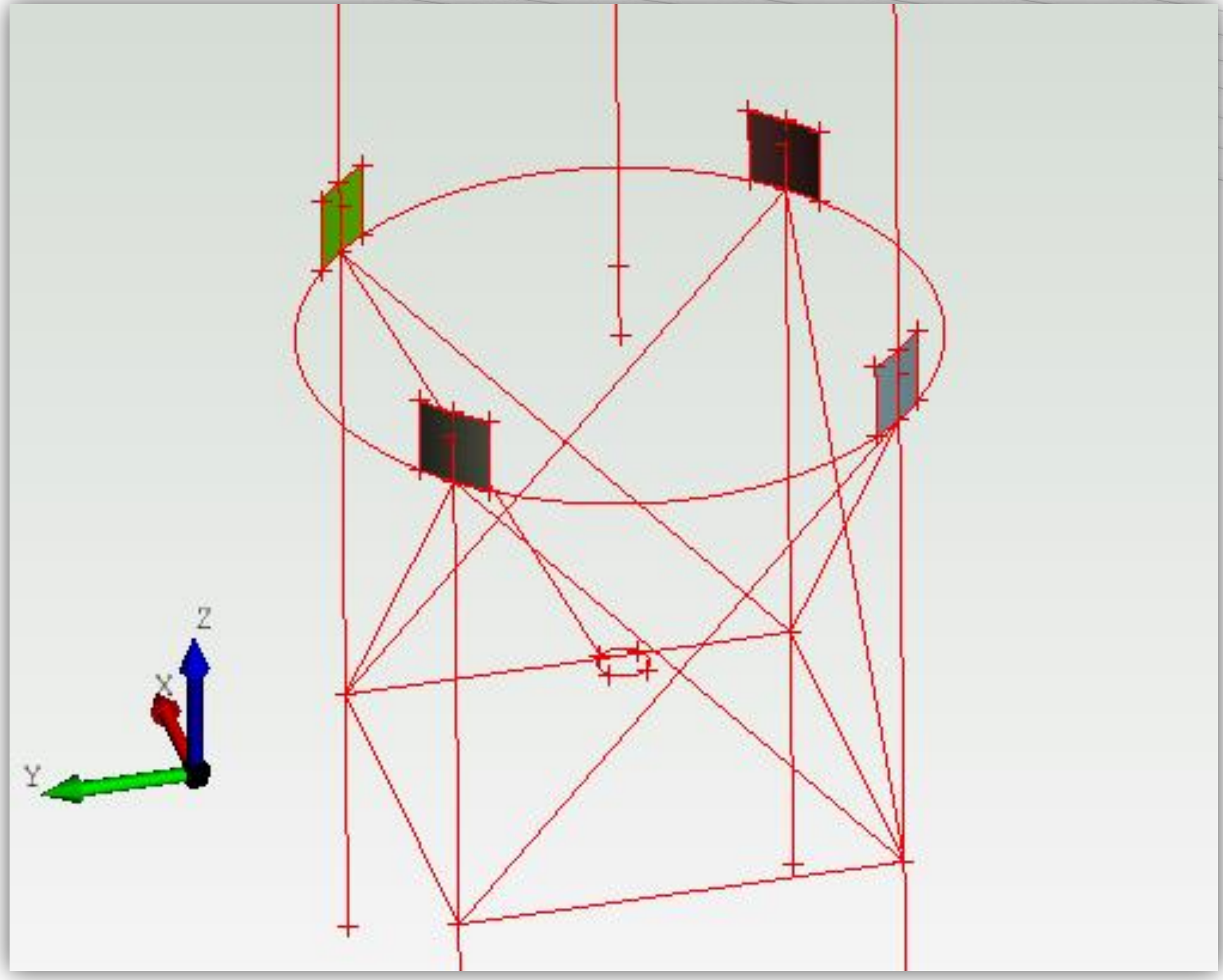
In order to define the stiffeners follow these steps:

1. Click on **Geometry** tab.
2. Click on **Polyline Surface** tab.
3. Select **By curves**.
4. Select surface boundaries.
5. Click on **OK**.





Surfaces



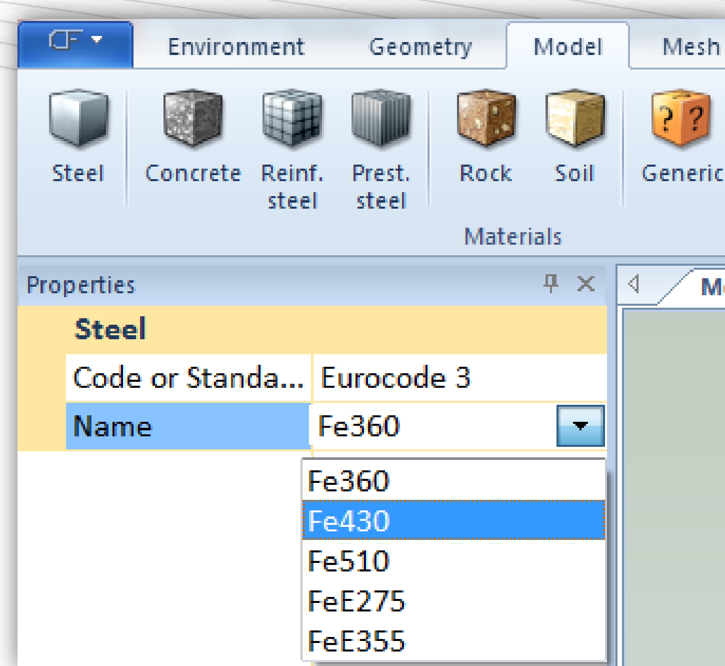
INGECIBER, S.A.

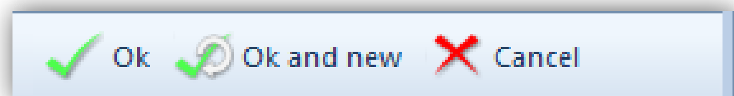
CivilFEM

Materials

We have to create materials for the silo.

1. Click on **Model** tab.
2. Click on **Steel** tab.
3. Select **Eurocode 3**.
4. Select **Fe 430** for silo.
5. Click on **OK and new**.
6. Select **Fe 510** for braces and stiffeners.
7. Click on **OK.**

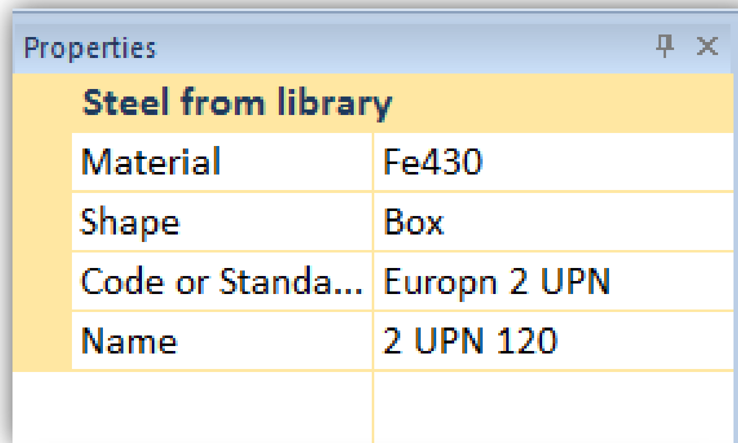


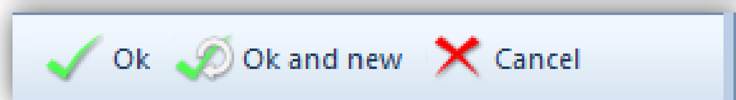


Sections

To create the **pilar** sections, follow these steps:

1. Click on **Model** tab.
2. Click on **Steel from library** tab.
3. Select **Fe 430.**
4. In shape select **Box**.
5. **Code** European 2 UPN
6. **Name** 2 UPN 120
7. Click on **OK**.





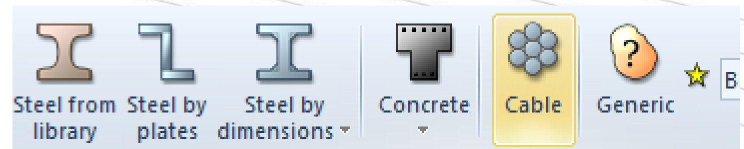
INGECIBER, S.A.

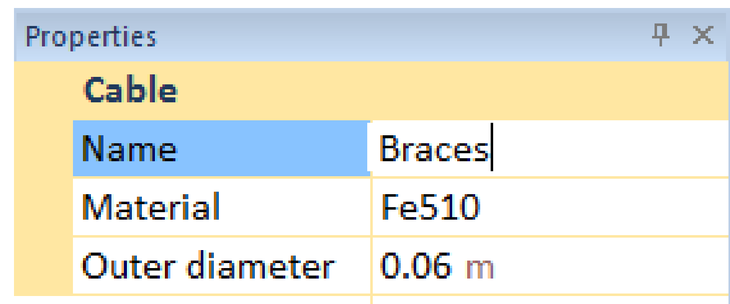
CivilFEM

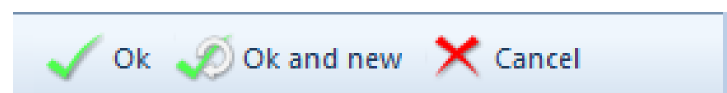
Sections

To create the **brace** sections, follow these steps:

1. Click on **Model** tab.
2. Click on **Steel from library** tab.
3. Select **Fe 430.**
4. In shape select **Box.**
5. **Code** European 2 UPN
6. **Name** 2 UPN 120
7. Click on **OK**.



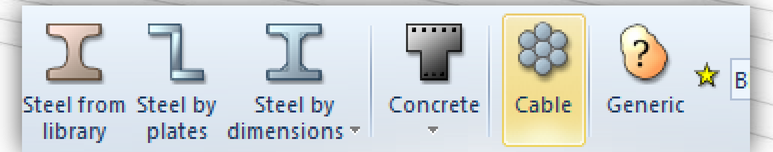


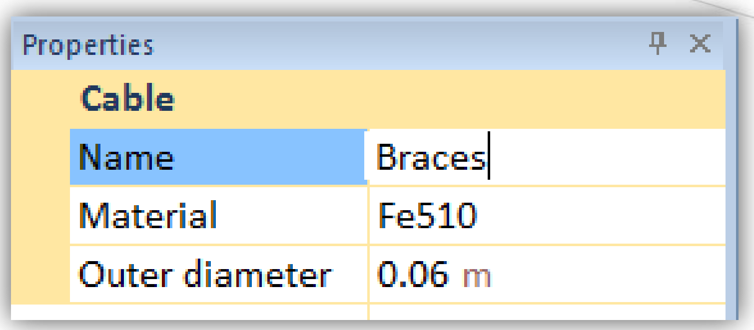


Sections

To create the **brace** sections, follow these steps:

1. Click on **Model** tab.
2. Click on **Steel from library** tab.
3. Select **Fe 430.**
4. In shape select **Box**.
5. **Code** European 2 UPN
6. **Name** 2 UPN 120
7. Click on **OK**.





Structural elements

Once we have defined all the geometry entities, materials and sections, we have to create the structural elements.

1. Click on **Mesh** tab.
2. Choose between **Beam**, **Truss** or **Shell**.

Choose Beam for pilars, Truss for braces and Shell for bodies of the silo, hooper and top.

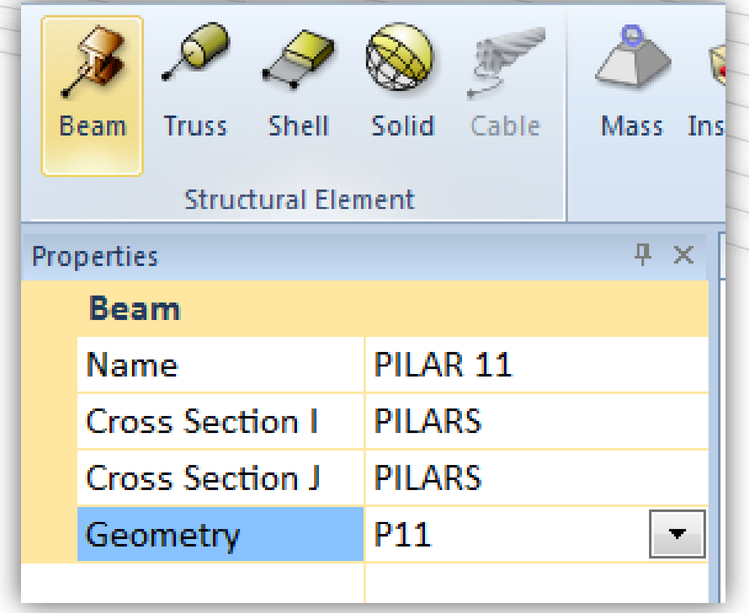
INGECIBER, S.A.

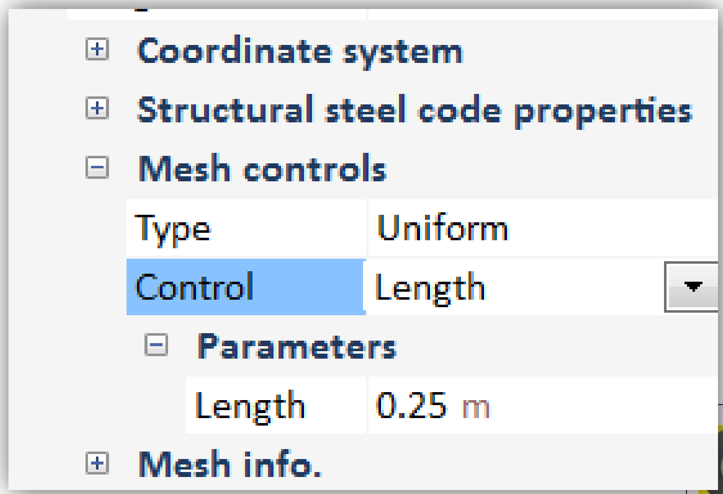
CivilFEM

Structural elements

Beams:

1. Click on **Beam**.
2. Select the **Section**.
3. Choose the **Geometry** to convert to structural element.
4. In **Properties > Mesh controls** select **Uniform** as **Type** and **Length** as **Control**
5. In **Parameters** enter as lenght 0,25 m.





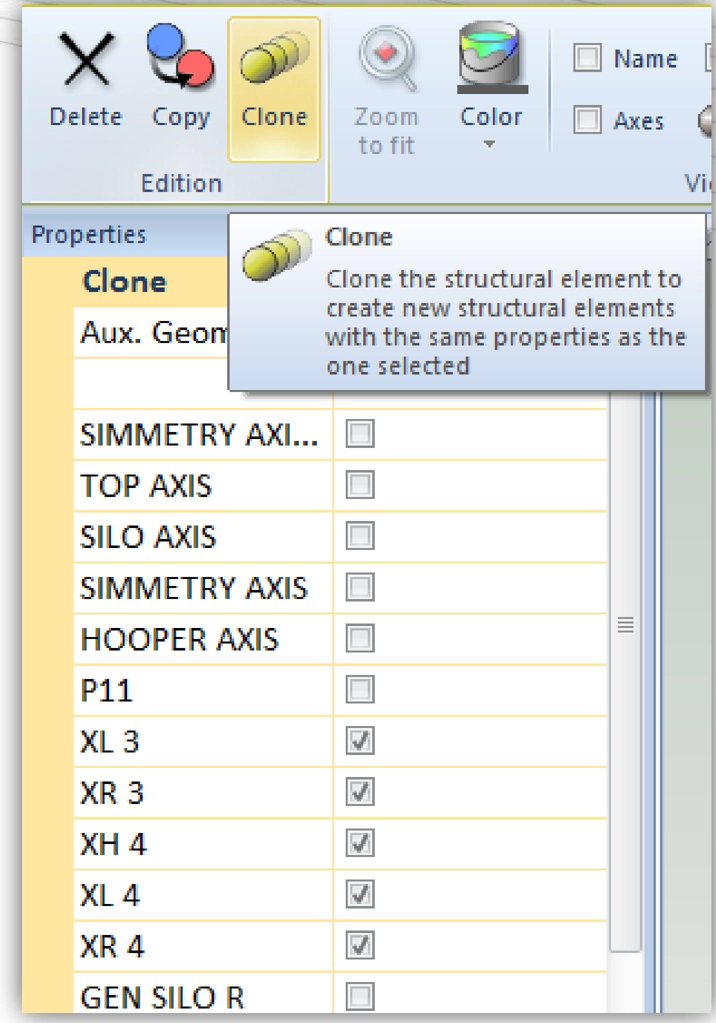
INGECIBER, S.A.

CivilFEM

Structural elements

Now we can clone the structural element and its properties instead of creating all beams one by one.

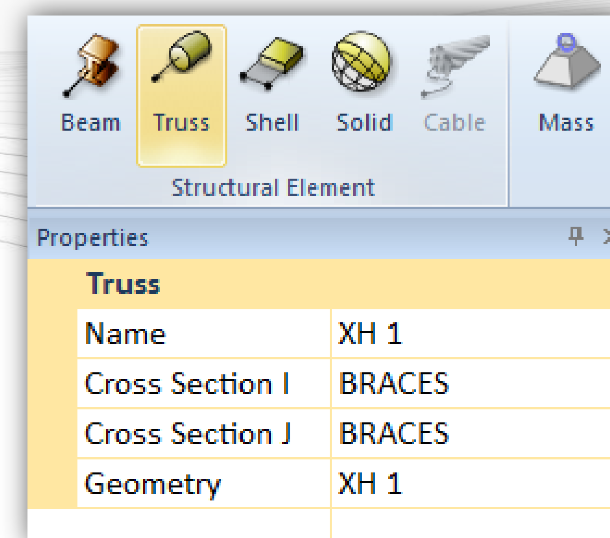
1. Click on the structural element.
2. Clik on **Clone**.
3. Select the geometry entities
4. Clik on **OK**.

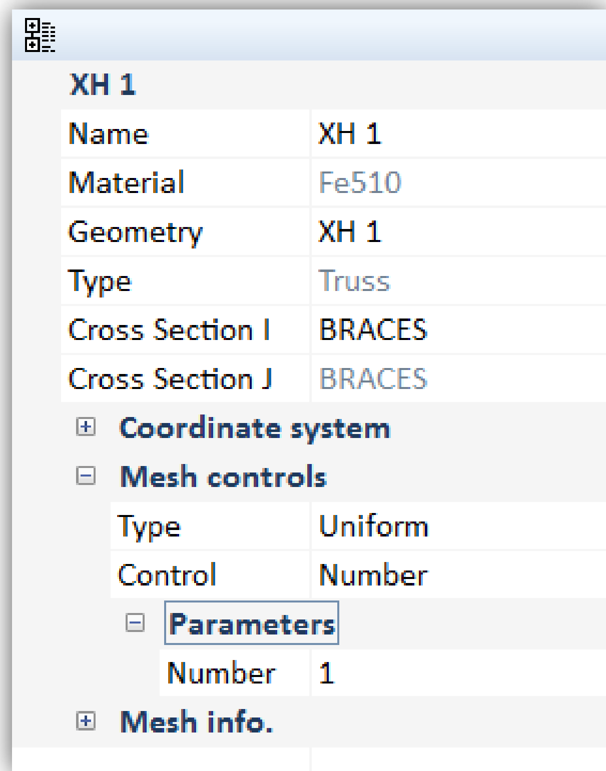


Structural elements

Truss:

1. Click on **Truss**.
2. Select the **Section**.
3. Choose the **Geometry** to convert in structural element.
4. In **Properties > Mesh controls** select **Uniform** as **Type** and **Number** as **Control.**
5. In **Parameters** enter 0,25 m as **Number**
6. **Clone** for all braces.





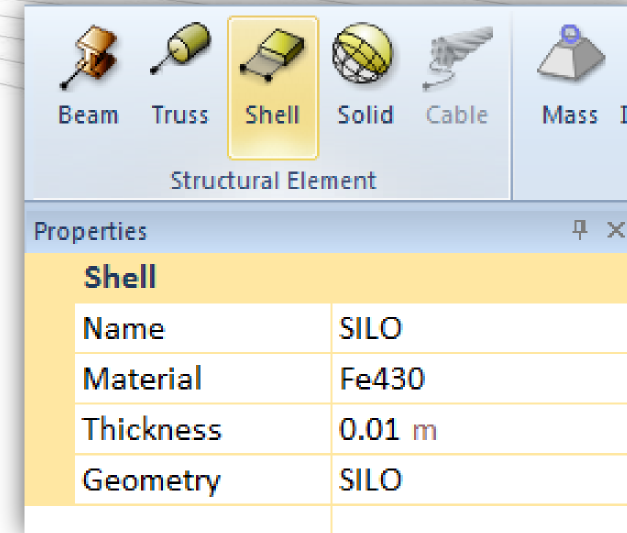
INGECIBER, S.A.

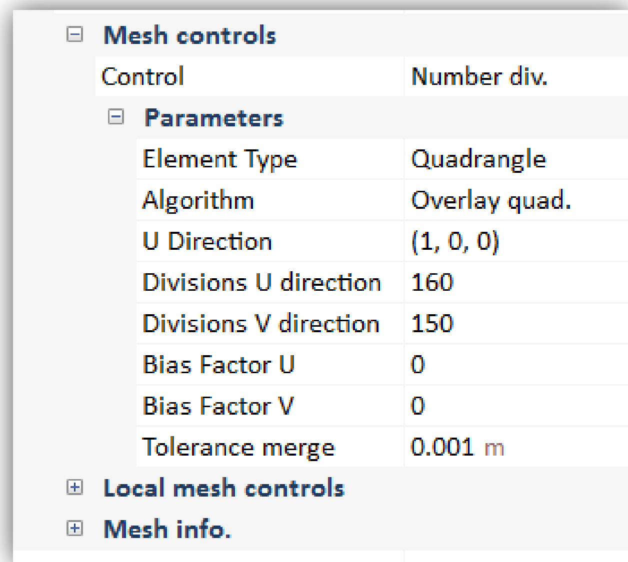
CivilFEM

Structural elements

Shell:

1. Click on **Shell**.
2. Select the **Material**.
3. Enter **Thinckness** value
4. In **Properties > Mesh controls** select **Number of divisions** as **Control.**
5. Select **Quandrangle** as **Element Type** .
6. Define **Divisions** on **U** and **V** directions as 160 and 150 respectively.
7. **Clone** for all braces.



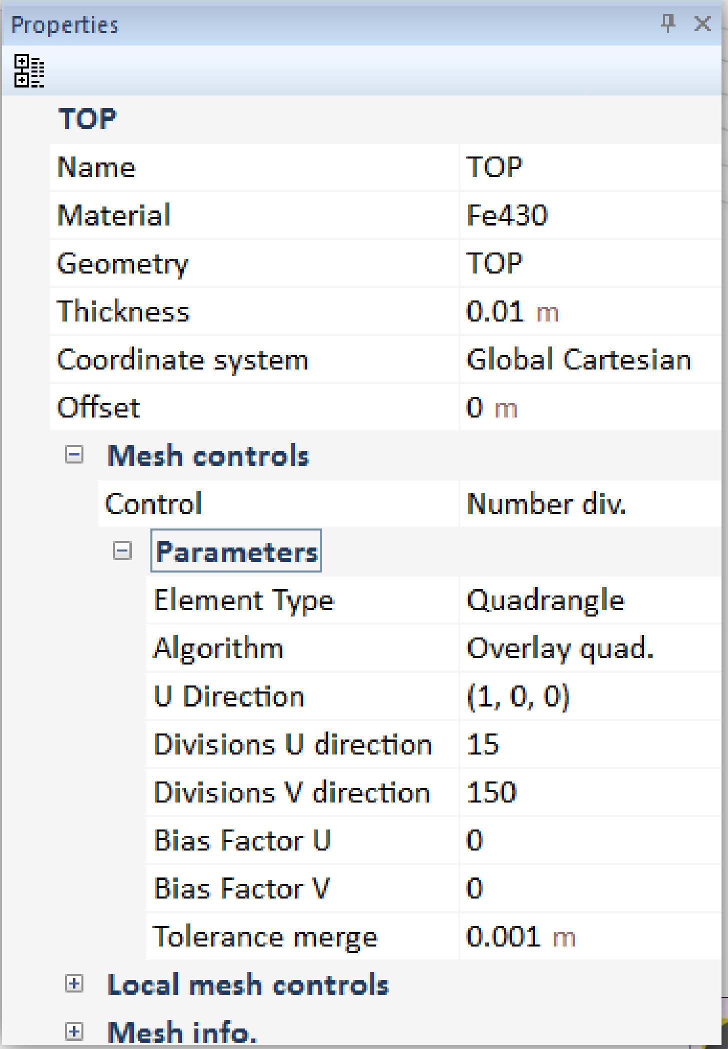


INGECIBER, S.A.

CivilFEM

Structural elements

For **Top** and **Hopper** shells, the procedure is the same: change the value of the **V Division** direction to 15 and **Direction U** to 150.

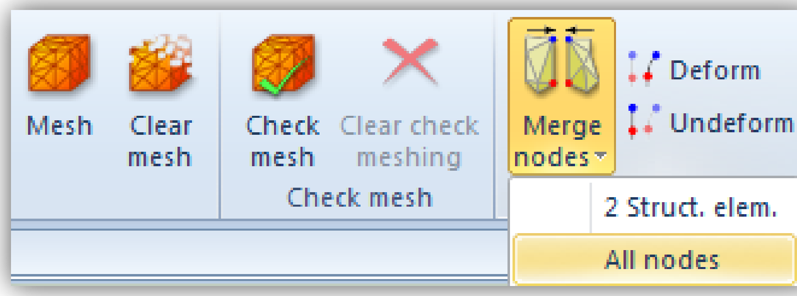


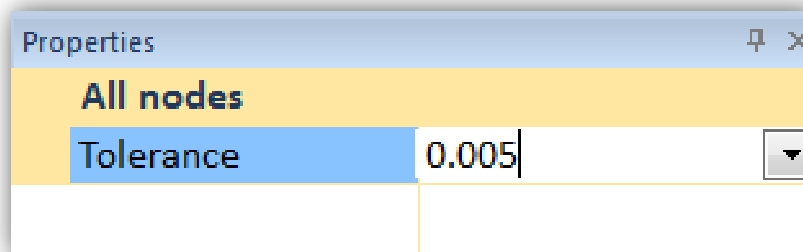
Mesh

1. Click on **Mesh** tab.
2. Click on **Mesh** again.

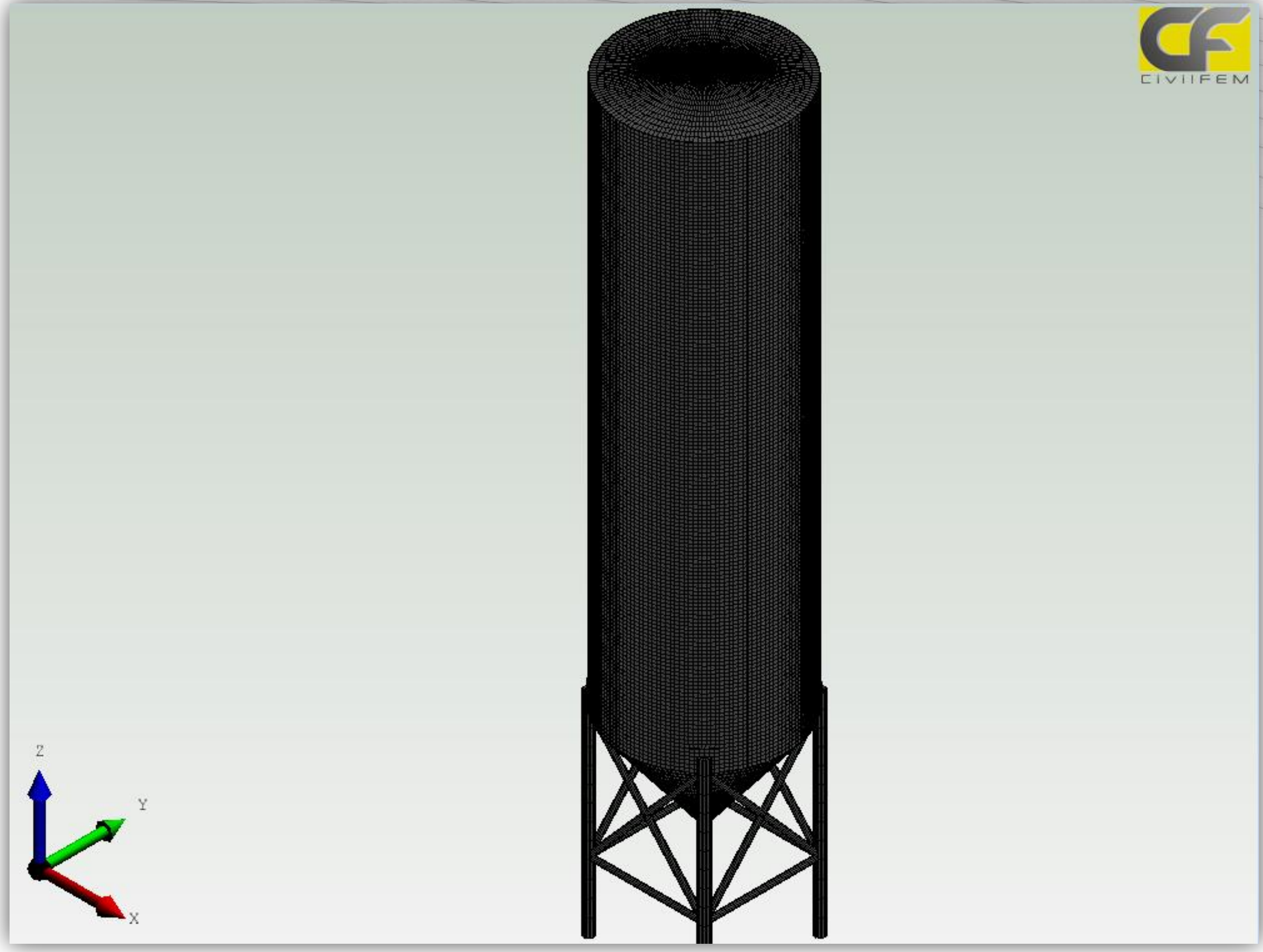
Once the model is meshed:

1. Click on **Merge** nodes tab.
2. Click on **All nodes**.
3. Enter 0.005 for **Tolerance**



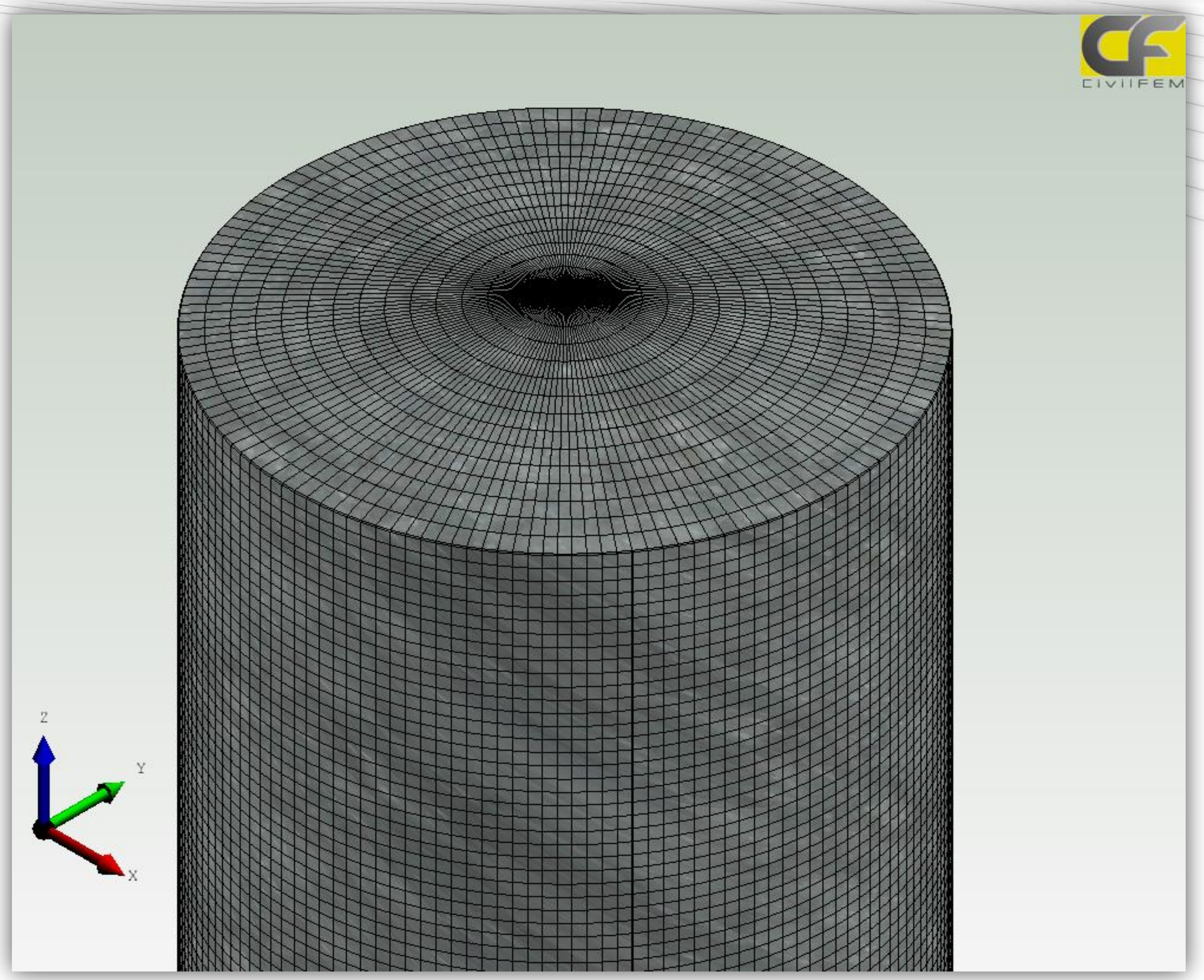


Mesh



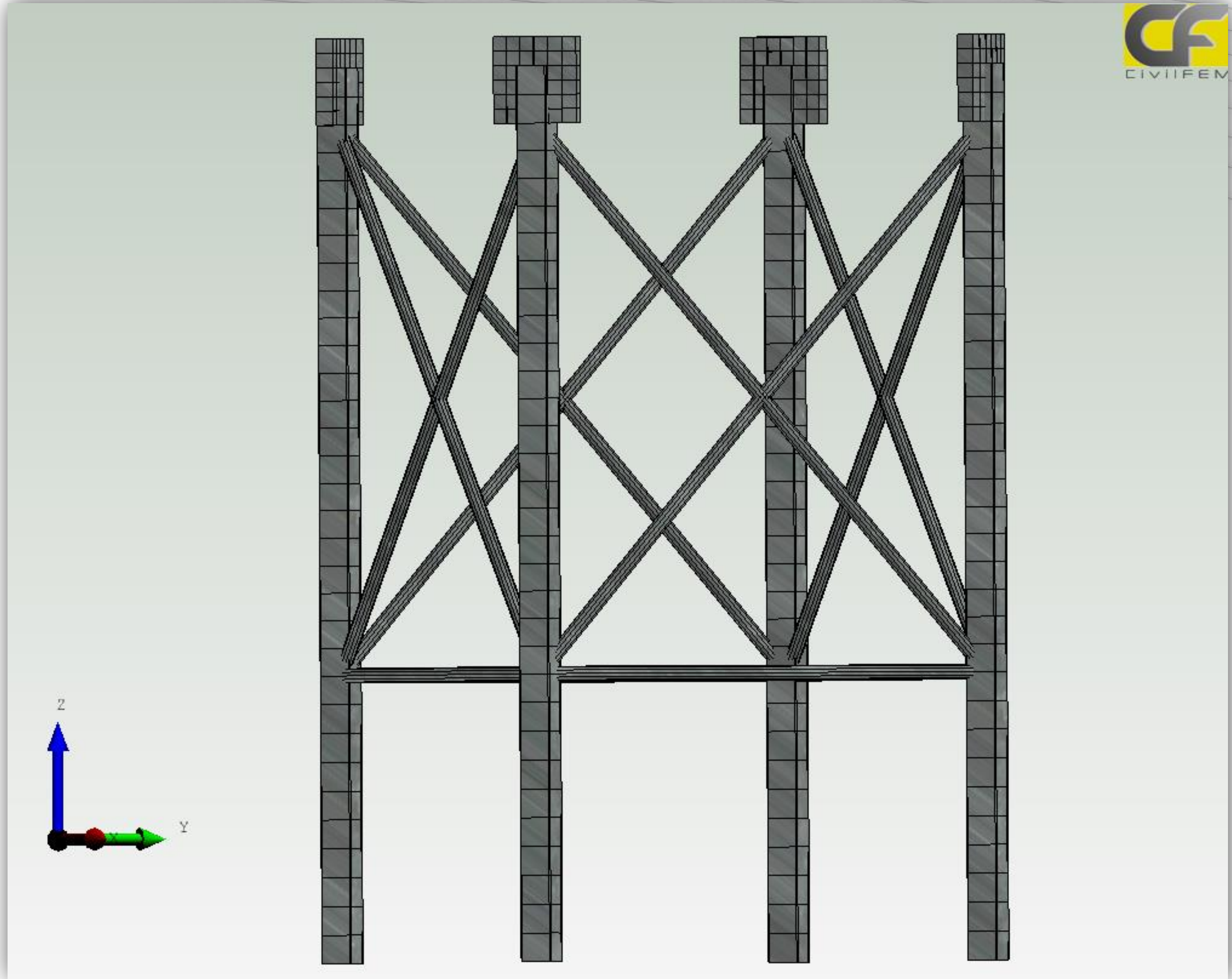
INGECIBER, S.A.

Mesh

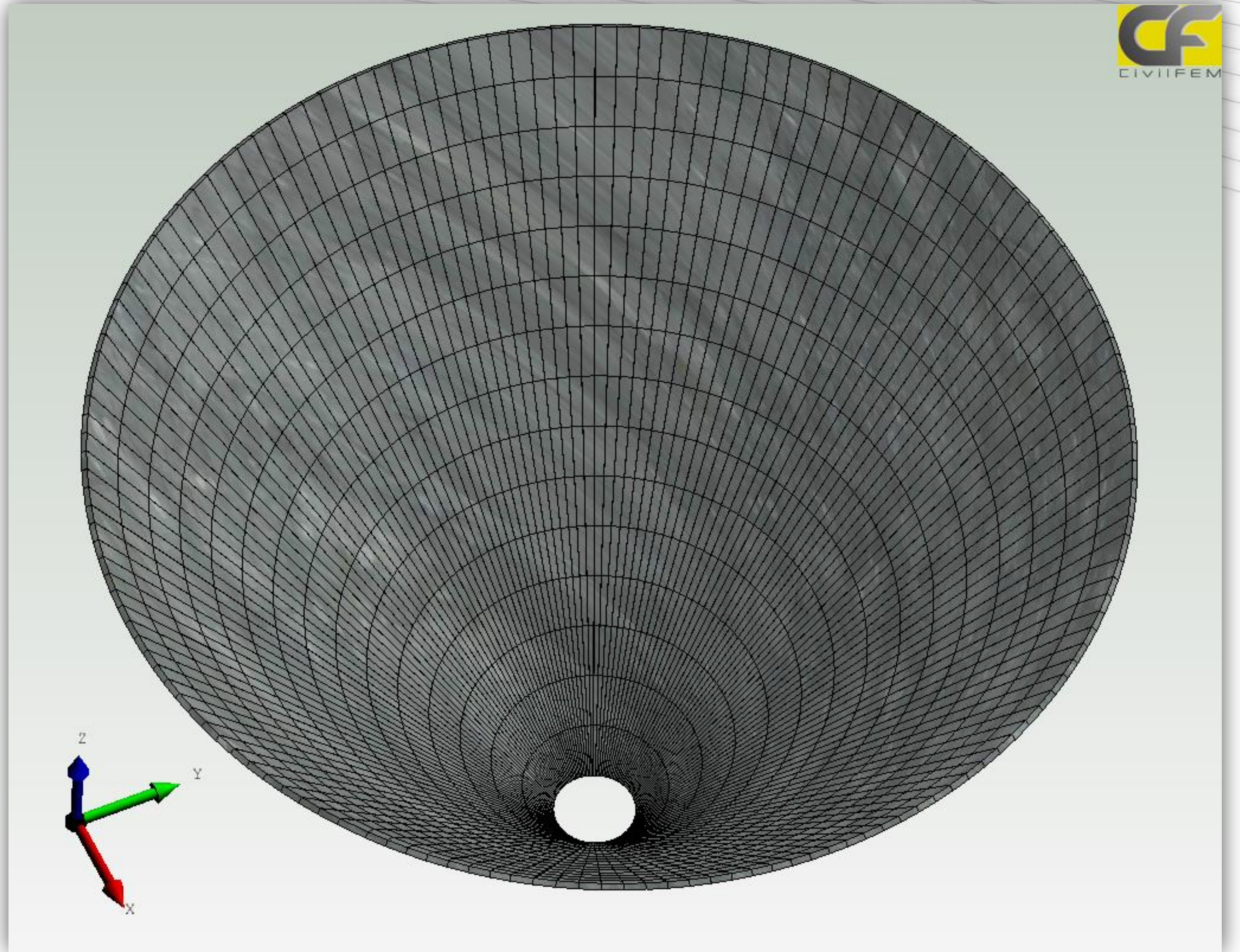


Mesh

Mesh



Mesh



INGECIBER, S.A.

CivilFEM

Loads

In this stage we are going to create the loads that will be applied to the silo.

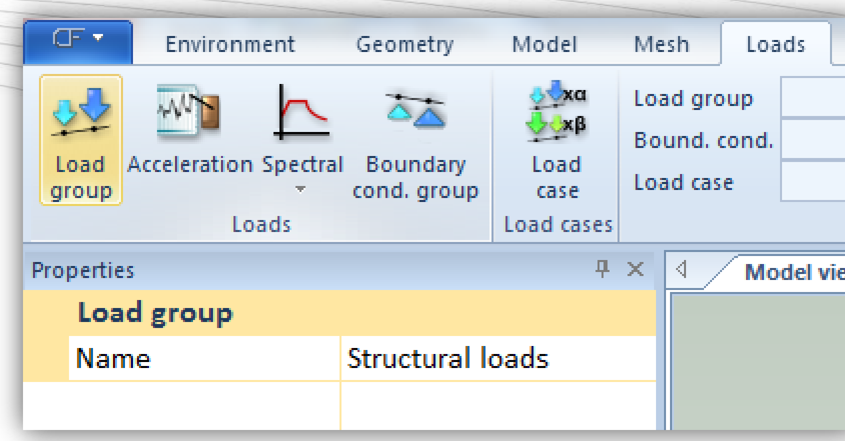
The loads to consider are: self weight, concrete weight, wind load (windward and leeward), and seismic load:

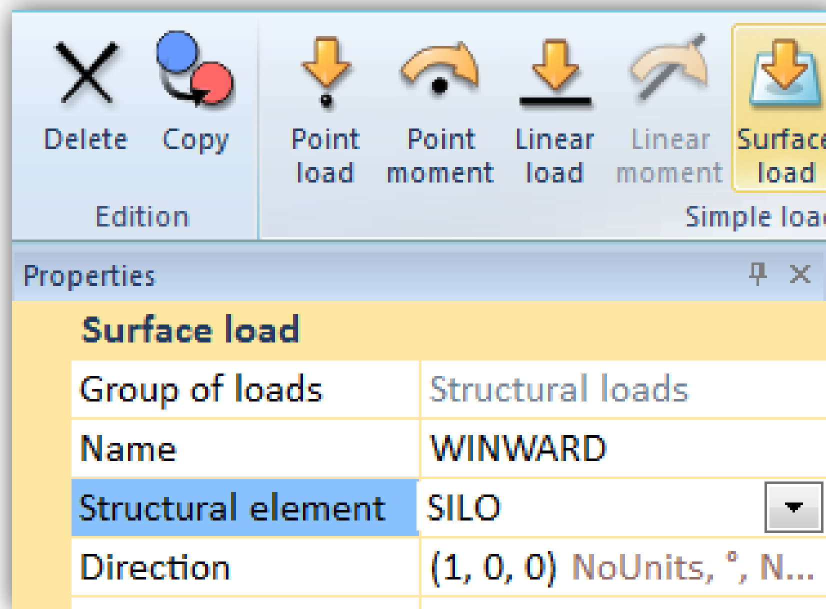
SW: Self weight
CW: Concrete Weight
W: Wind (Windward and Leeward)
SHP: Static horizontal pressure
DHP: Dynamic horizontal pressure (DHP=1,40·SPH)
S: Seismic load

Wind load

We create the wind load.

1. Click on **Loads** tab.
2. Click on **Load group** tab.
3. Click on **OK**.
4. Click on **Surface load** tab.
5. Put a **Name**.
6. Select the structural element where the load will be applied.
7. Click on **OK**.





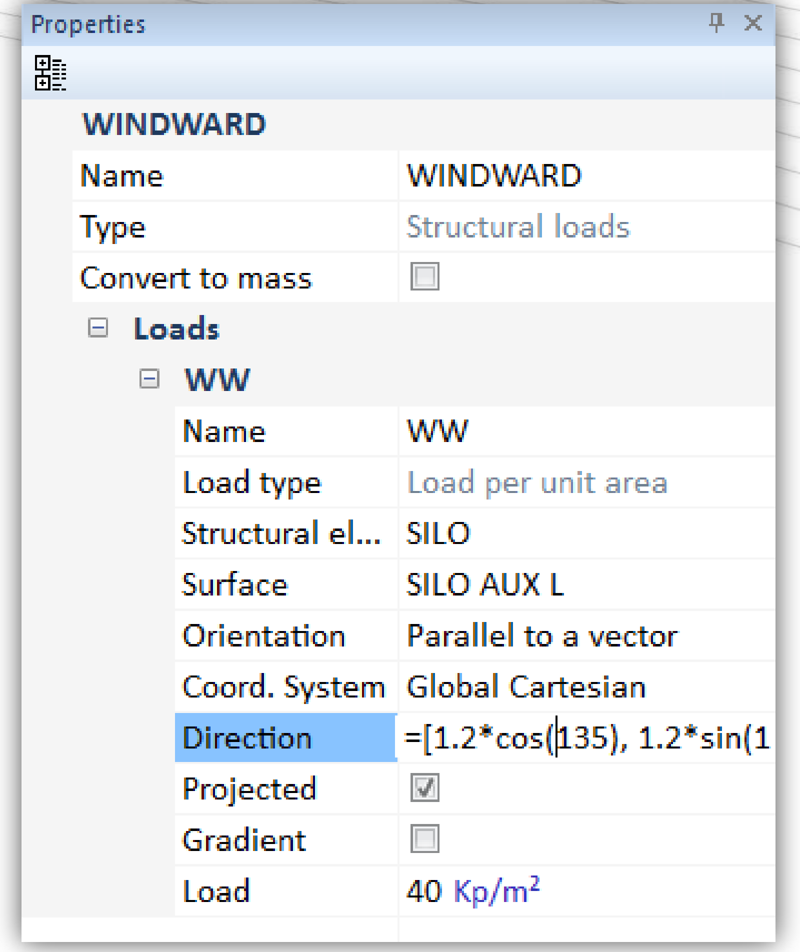
INGECIBER, S.A.

CivilFEM

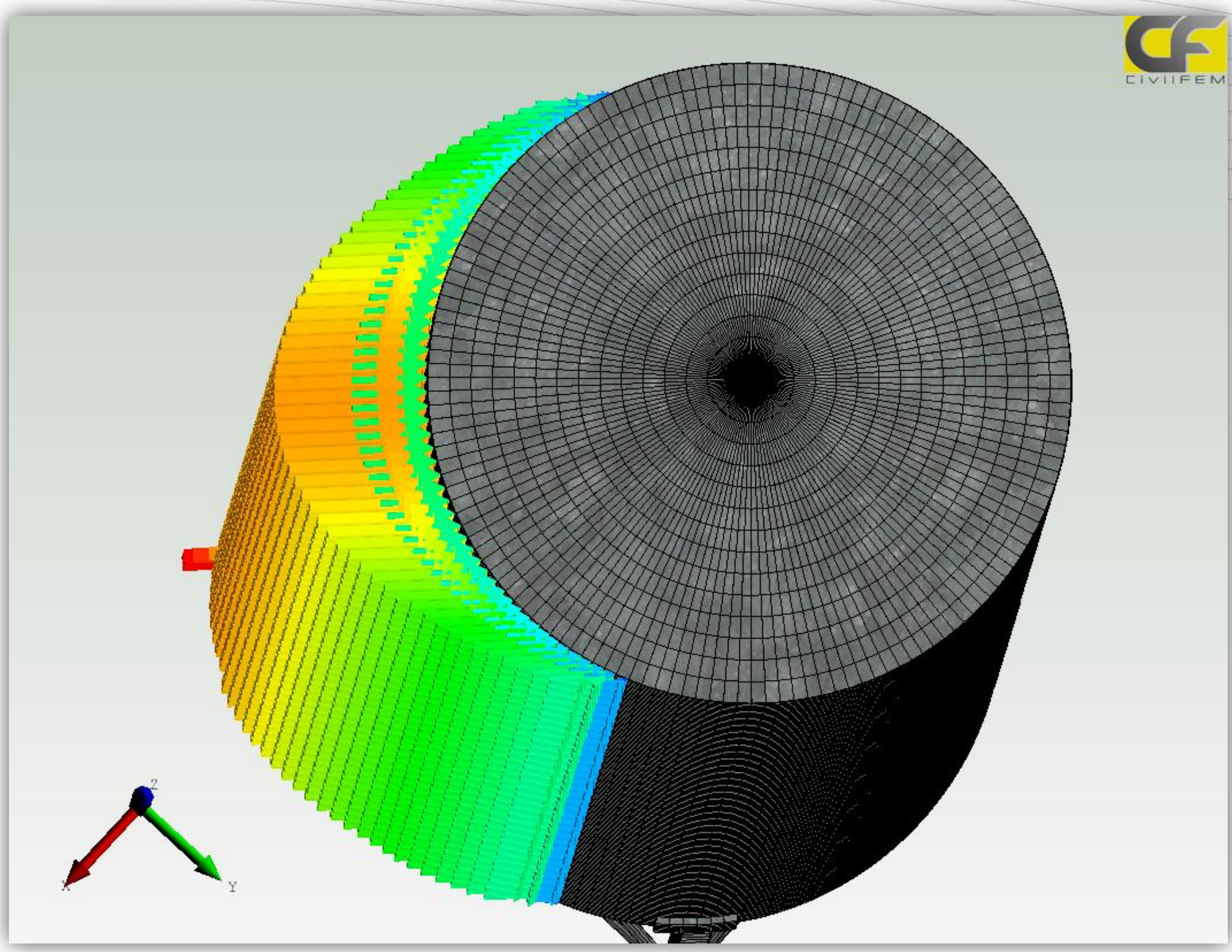
Loads

We now edit the load **Properties**:

1. Display **Loads**.
2. Select SILO AUX L as **Surface**.
3. In orientation, select **Parallel to a vector**.
4. In Direction enter **[1,2·cos(135), 1,2·sin(135), 0]**
5. Check **Projected** box.
6. And finally enter the value of the load, in this case 40 kp/m2.



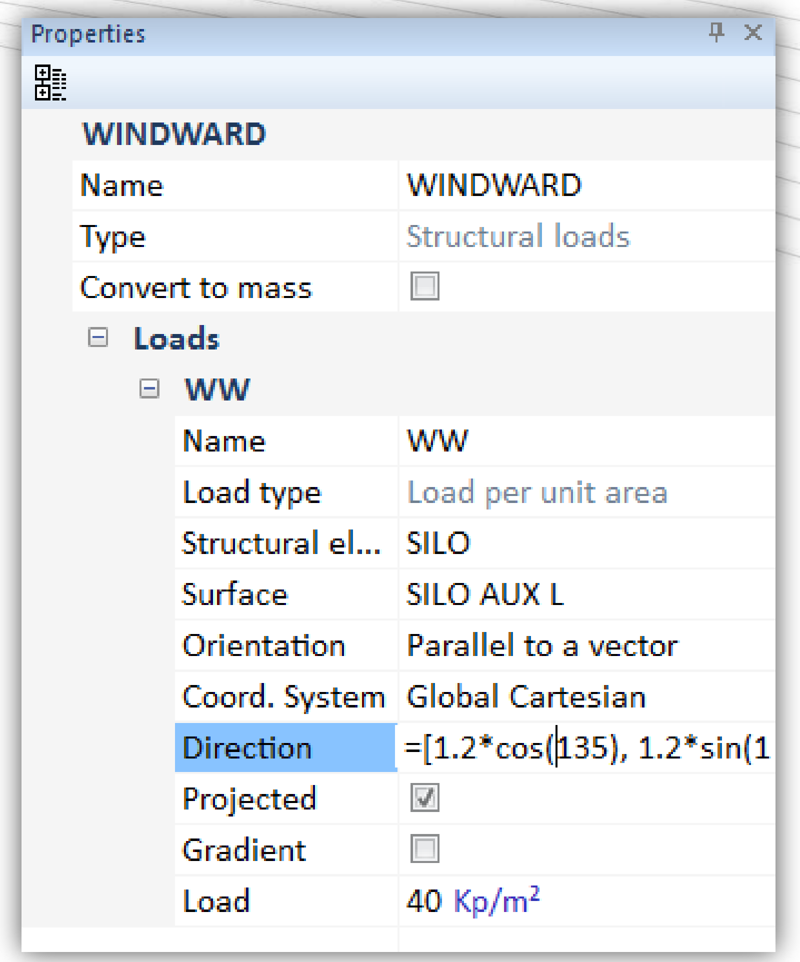
Windward load



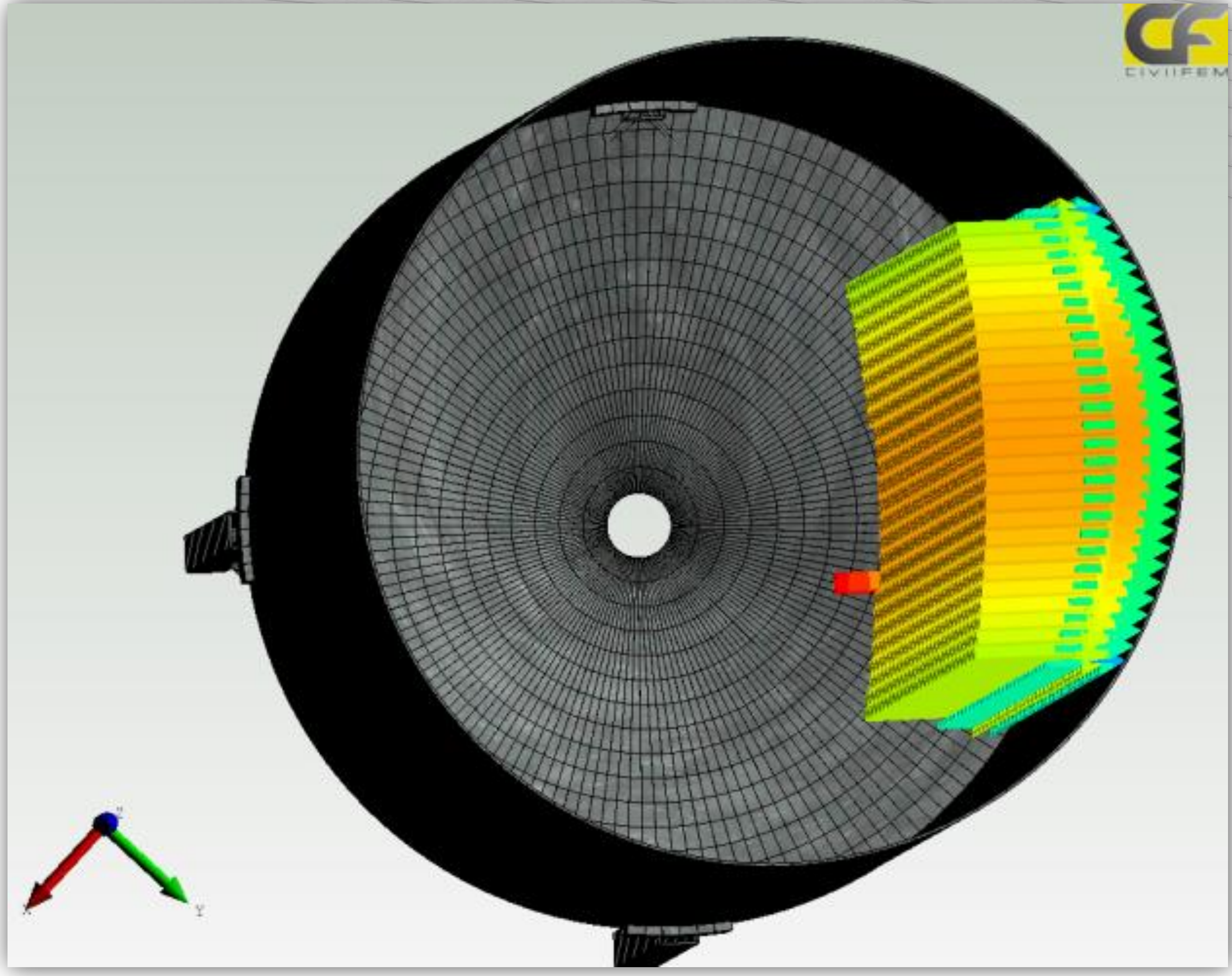
Leeward load

We define the Leeward load in the same way, but in this case, we have to select the other side of the silo.

1. Display **Loads**.
2. Select SILO AUX R. as **Surface**
3. In orientation, select **Parallel to a vector**.
4. In Direction enter **[1,2·cos(135), 1,2·sin(135), 0]**
5. Check **Projected** box.
6. And finally enter the value of the load, in this case 20 kp/m2.



Leeward load

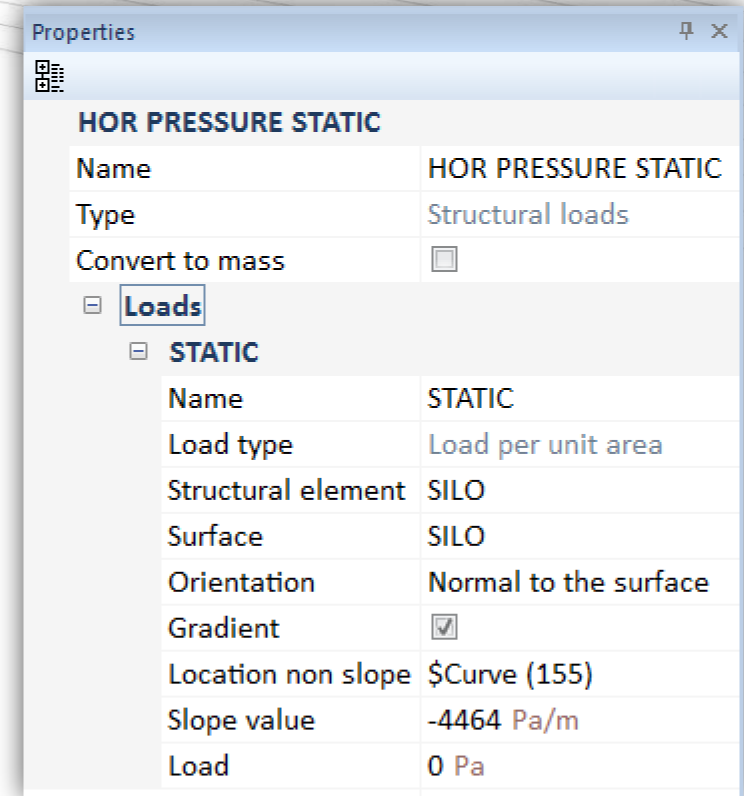


INGECIBER, S.A.

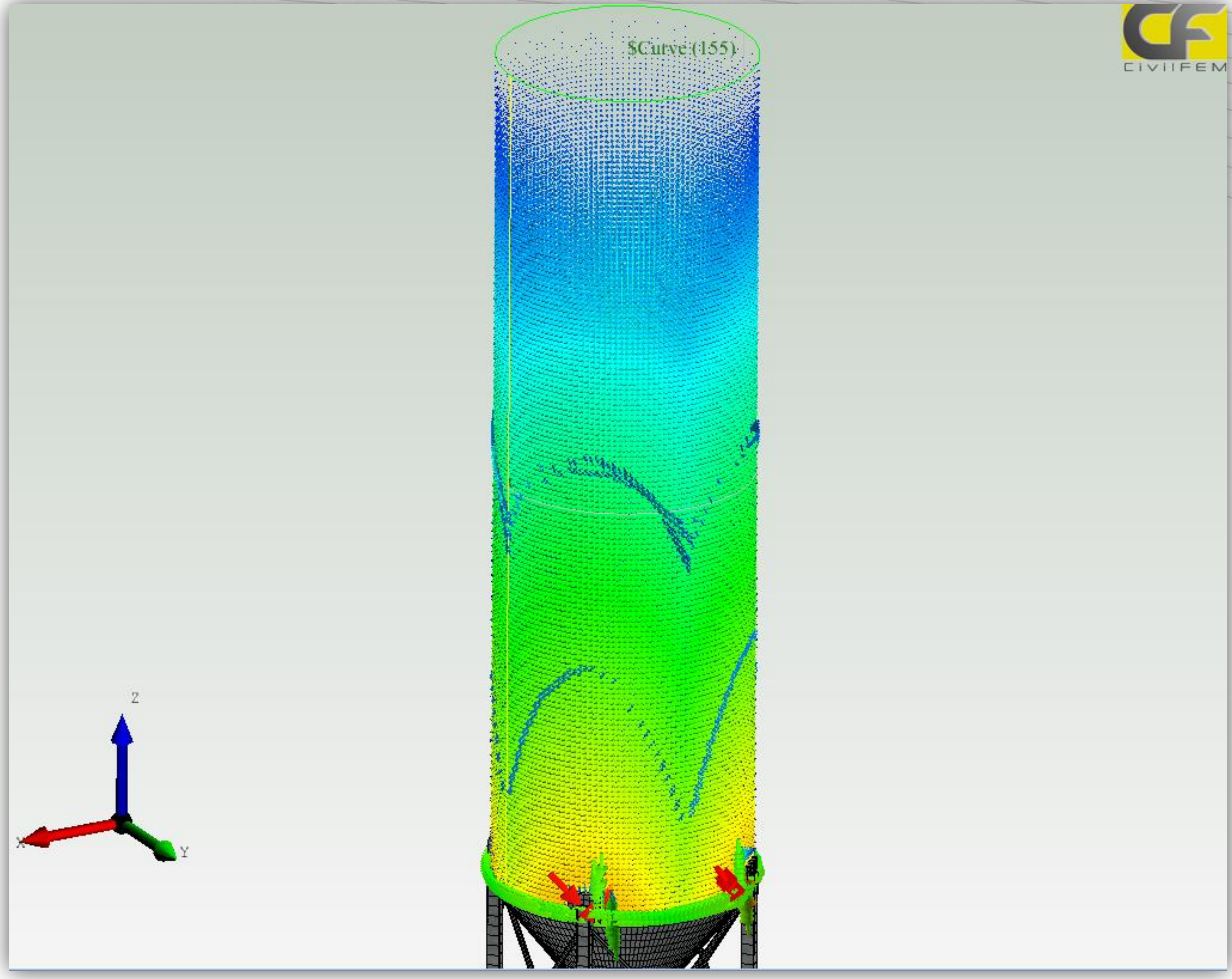
Static horizontal pressure

This is also a surface load applied inside of silo.

1. Create the **Superficial load**.
2. Display **Properties**.
3. Select **Silo** as **Structural element** and **Surface**.
4. Select **Normal to the surface** as **Orientation**.
5. Check **Gradient** box.
6. Choose the upper auxiliar curve of the silo in **Location non slope**.
7. Enter -4464 Pa/m as **Value**.



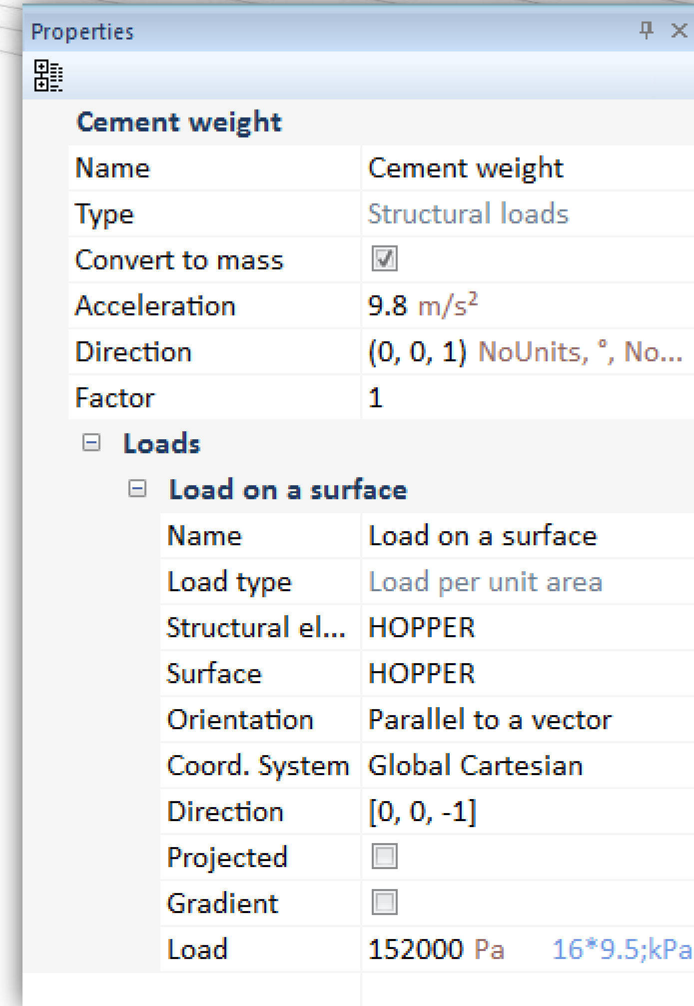
Static horizontal pressure



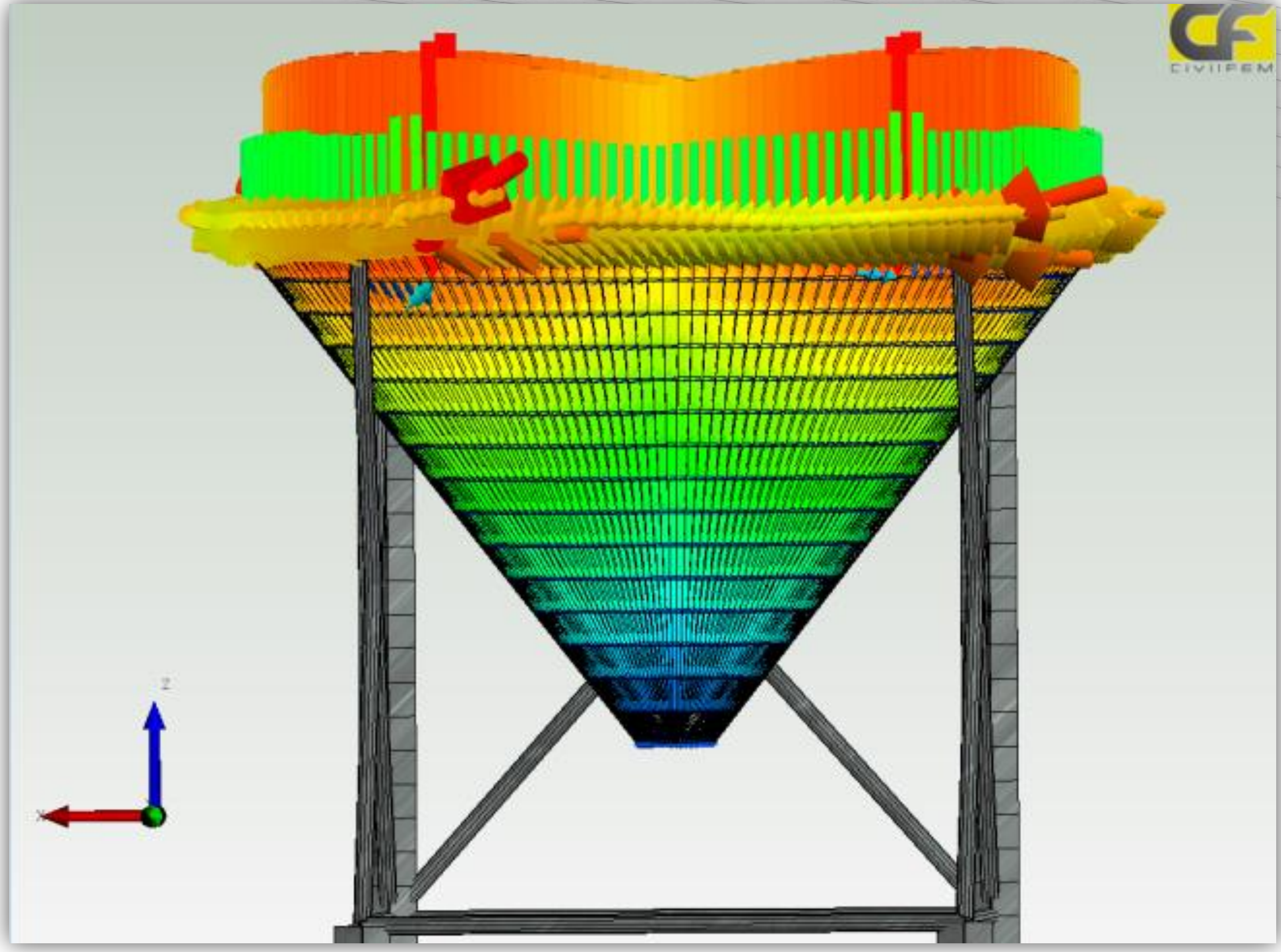
Cement load

Once again, create a **Surface load** and go to **Properties**:

1. Check **Convert to mass** box.
2. Display **Loads.**
3. Select Hopper as **Surface**.
4. In **Orientation**, select **Parallel to a vector**.
5. **Direction** [0, 0 -1]
6. Enter the **Load**, 152 kPa.



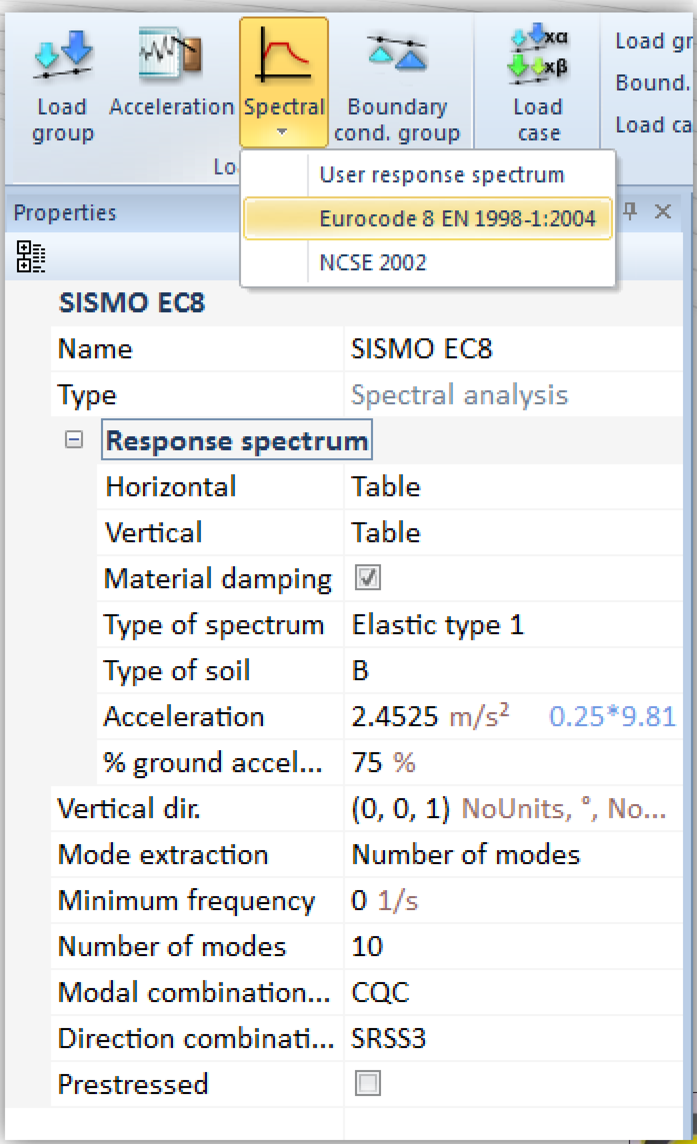
Cement load



Seism

To create the seismic load according to Eurocode 8, follow these steps:

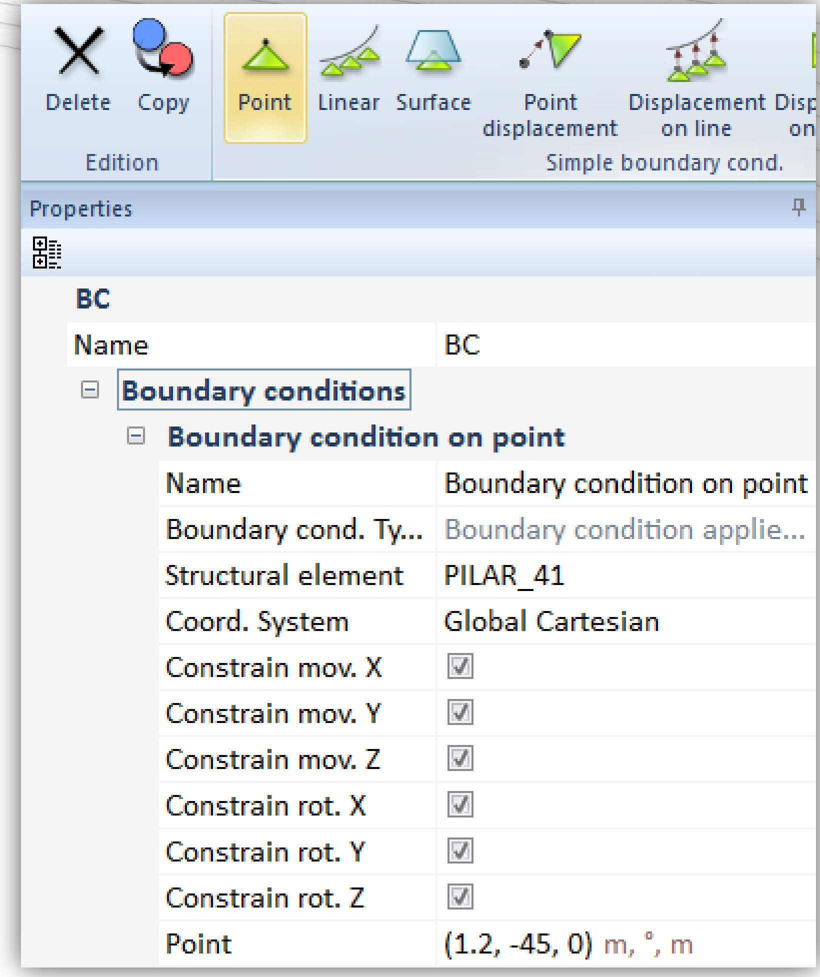
1. Click on **Loads** tab.
2. Click on **Spectral** tab.
3. Click on **Eurcocode 8**.
4. Click on **OK**.
5. Display **Properties.**
6. Select **Elastic type 1** as **Type of spectrum**.
7. Choose **Type of soil** B.
8. In **Acceleration**, enter 0,25·9,81.
9. **Number of modes**: 10



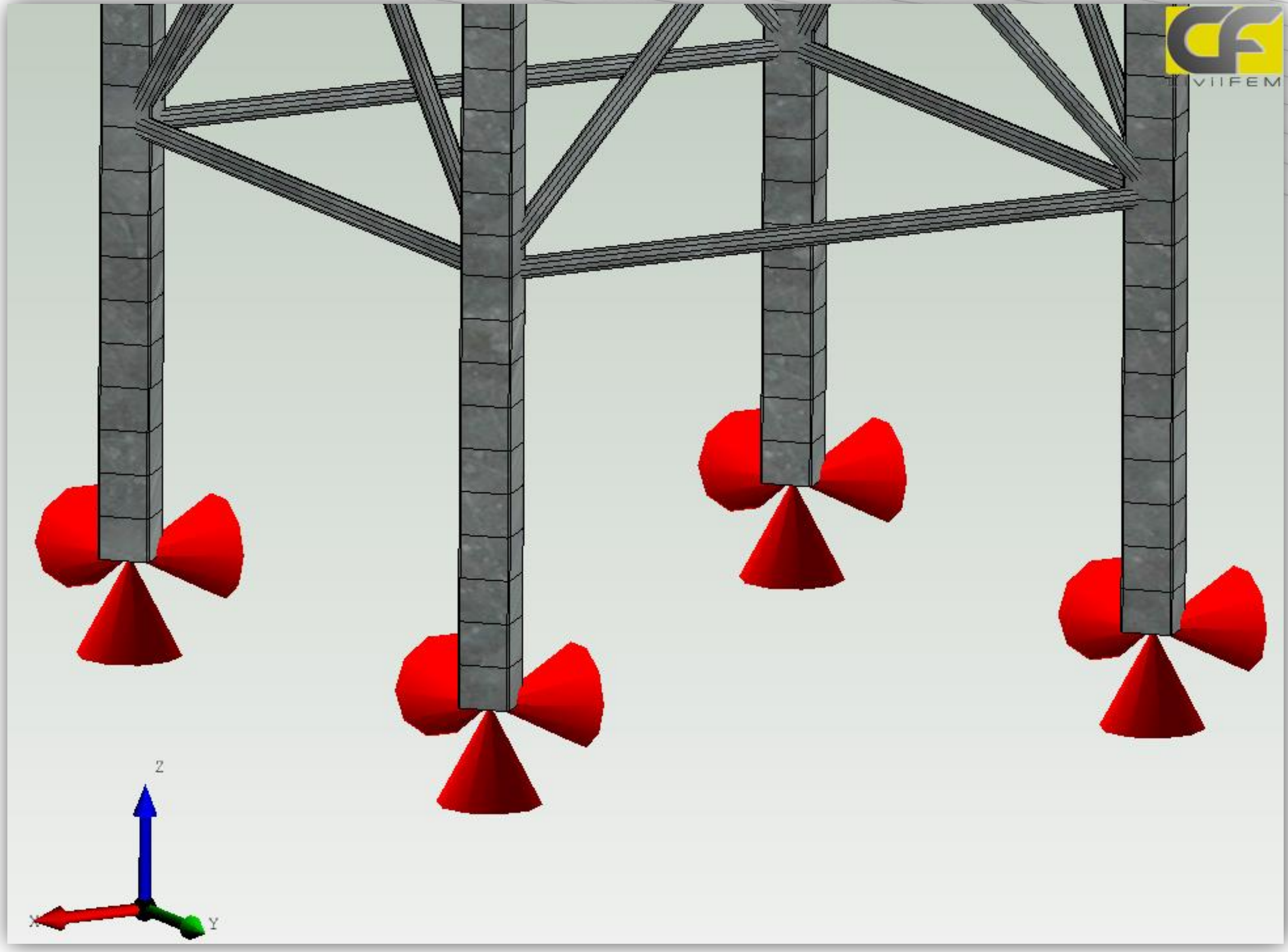
Boundary Conditions

It is essential to establish boundary conditions in the model.

1. Click on **Loads** tab.
2. Click on **Boundary conditons** tab.
3. Click on **OK.**
4. Display **Properties**.
5. Select a **Structural element**: Pilars.
6. **Constrain** all movements and rotations.
7. Repeat for all pilars.



Boundary Conditions



INGECIBER, S.A.

CivilFEM

Load cases

The last step before solving is creating the load cases.

We will have 4 load cases:

Self weight + seismic load: SW + S

Full static + seismic load : SW + CW + SHP + S

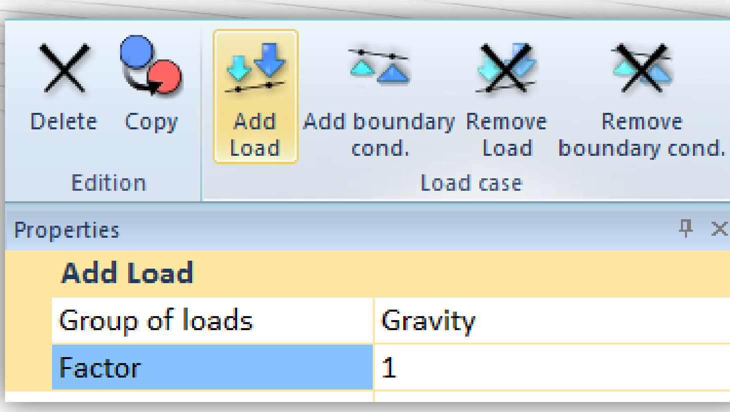
Discharge + seismic load : SW + CW + 1,40·SHP + S

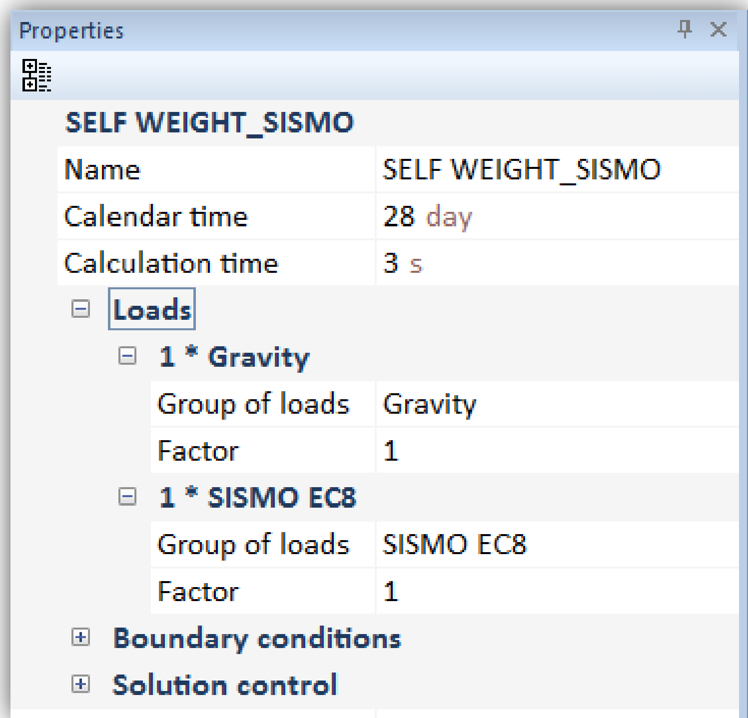
Wind: 1,35·SW+1,35·CW+1,5SHP+1,5 W

Load cases

Follow these steps:

1. Click on **Loads** tab.
2. Click on **Load cases** tab.
3. Click on **OK**.
4. Display **Properties**.
5. Click on **Add load**.
6. Choose **Gravity** as self weight.
7. **Factor:** 1, 1.35, 1.50, etc.
8. Click on **OK**.
9. Repeat for all load cases.





Load cases

Properties

FULL STATIC_SEISM	
Name	FULL STATIC_SEISM
Calendar time	28 day
Calculation time	2 s
Loads	
1 * Gravity	
Group of loads	Gravity
Factor	1
1 * SISMO EC8	
Group of loads	SISMO EC8
Factor	1
1 * HOR PRESSURE STATIC	
Group of loads	HOR PRESSURE STATIC
Factor	1
1 * HOPPER LOAD	
Group of loads	HOPPER LOAD
Factor	1
1 * Cement weight	
Group of loads	Cement weight
Factor	1
Boundary conditions	
Solution control	

Properties

DISCHARGE_SISMO	
Name	DISCHARGE_SISMO
Calendar time	28 day
Calculation time	1 s
Loads	
1 * Gravity	
Group of loads	Gravity
Factor	1
1 * HOPPER LOAD	
Group of loads	HOPPER LOAD
Factor	1
1.4 * HOR PRESSURE STATIC	
Group of loads	HOR PRESSURE STATIC
Factor	1.4 1.4
1 * SISMO EC8	
Group of loads	SISMO EC8
Factor	1
1 * Cement weight	
Group of loads	Cement weight
Factor	1
Boundary conditions	
Solution control	

Properties

Wind	
Name	Wind
Calendar time	28 day
Calculation time	5 s
Loads	
1.35 * Cement weight	
Group of loads	Cement weight
Factor	1.35
1.35 * Gravity	
Group of loads	Gravity
Factor	1.35
1.5 * LEEWARD	
Group of loads	LEEWARD
Factor	1.5
1.5 * WINDWARD	
Group of loads	WINDWARD
Factor	1.5
Boundary conditions	
Solution control	

Solving

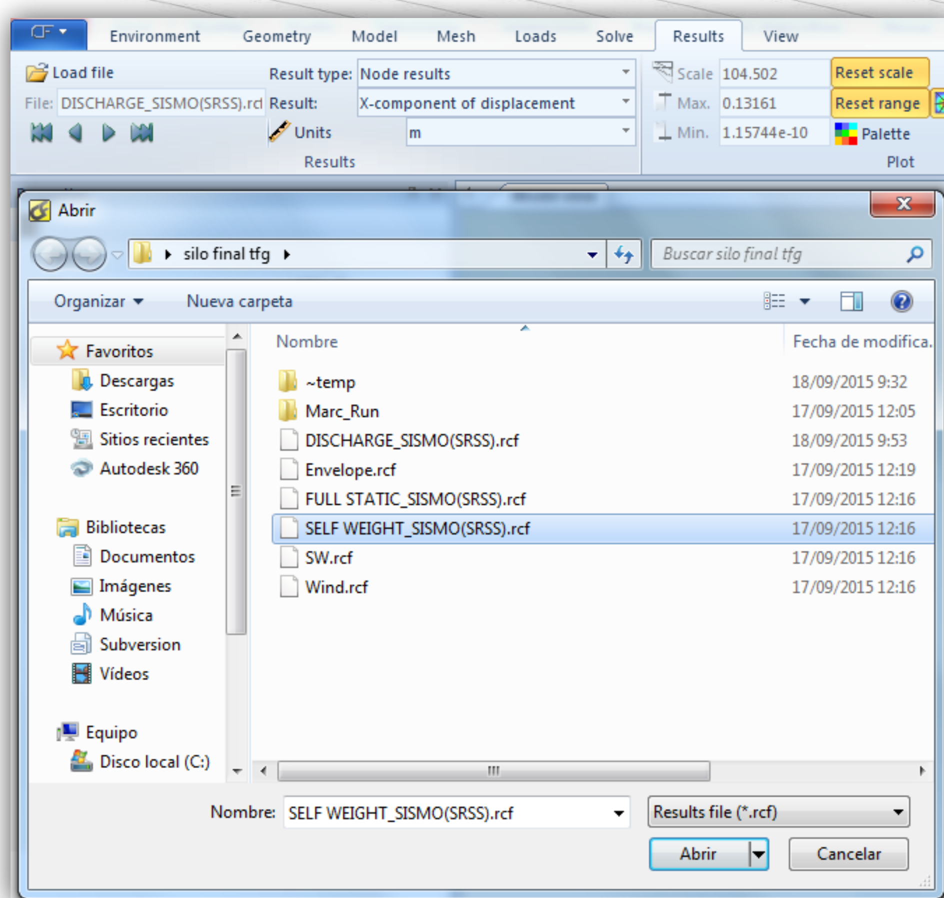
Finally, we have to solve the model.

1. Click **Solve** tab.
2. Click on **Start** tab.

Results

Once results have been obtained, we can load them into CivilFEM:

1. Click on **Results** tab.
2. Click on **Load file**.
3. Choose the file path and select the file.
4. Click on **Open**.

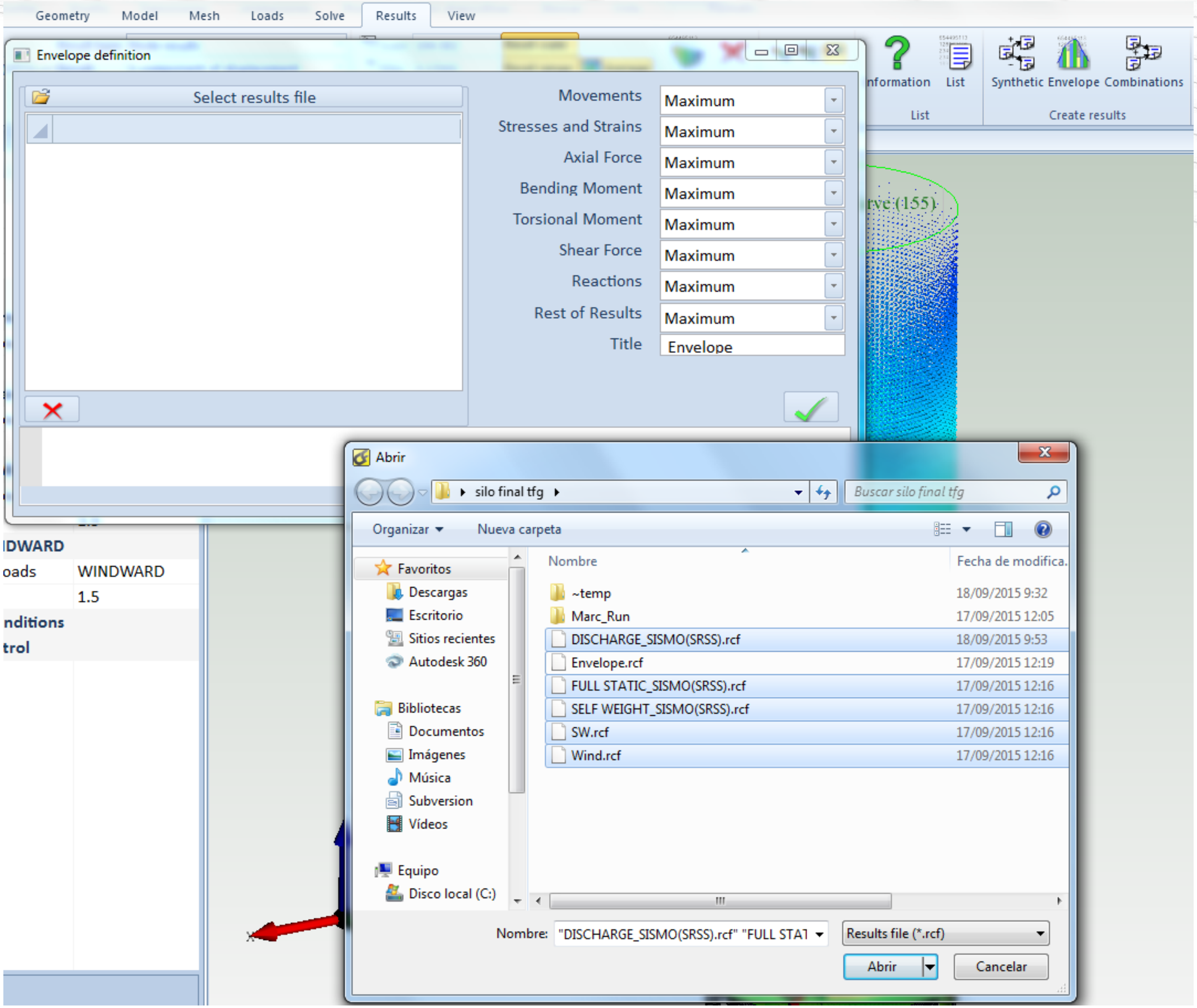


Envelope

We can review the result files one by one or create an **Envelope** that includes all results of all load cases. We can choose if we want maximum or minimum values.

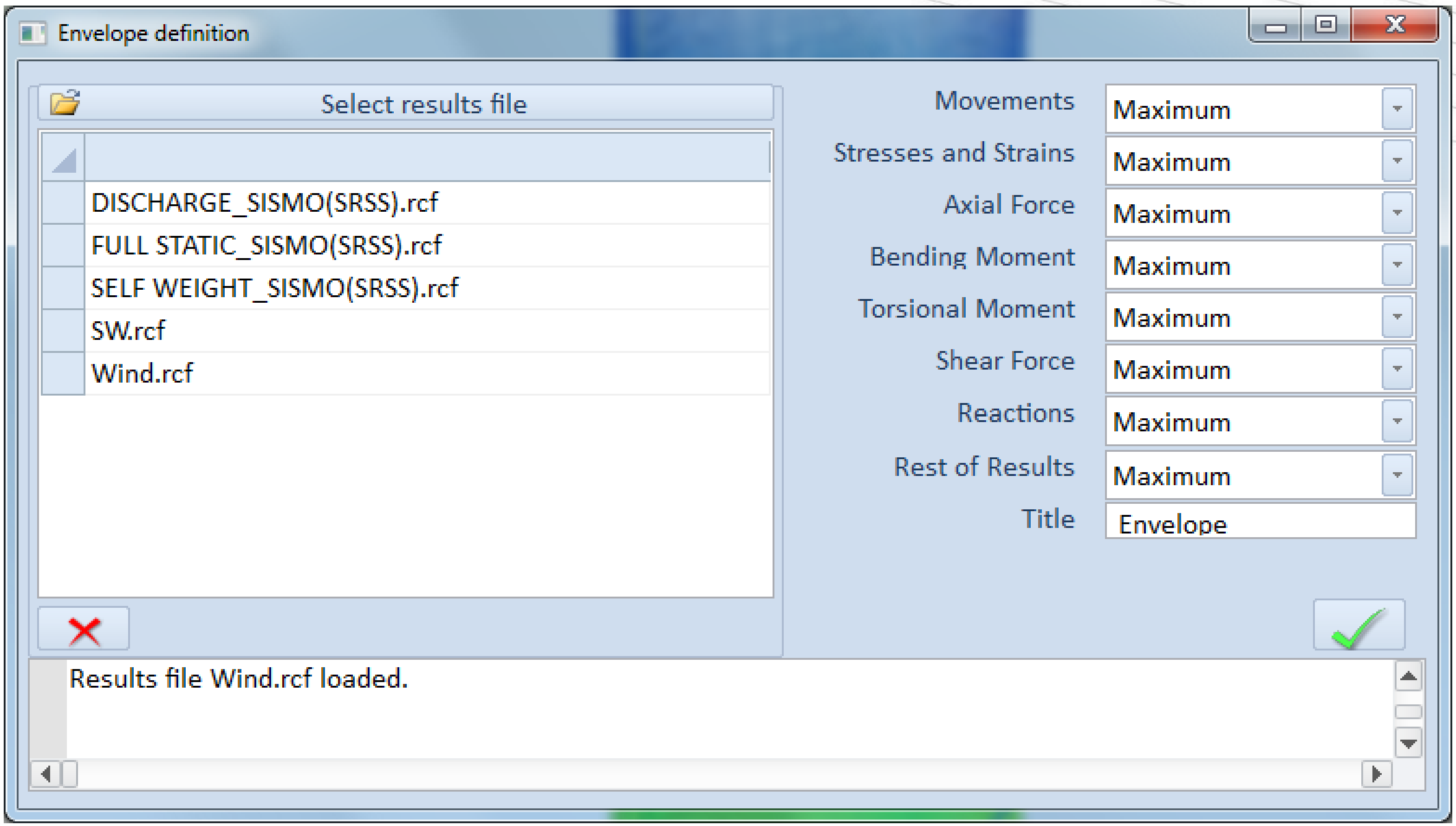
1. Click on **Results** tab.
2. Click on **Envelope** tab.
3. Cilck on **Select results file**.
4. Choose the path of the files and select all results files.
5. Click on **Open**.
6. Click on the Green check.
7. Finally, load the envelope file like in the previous slide.

Envelope

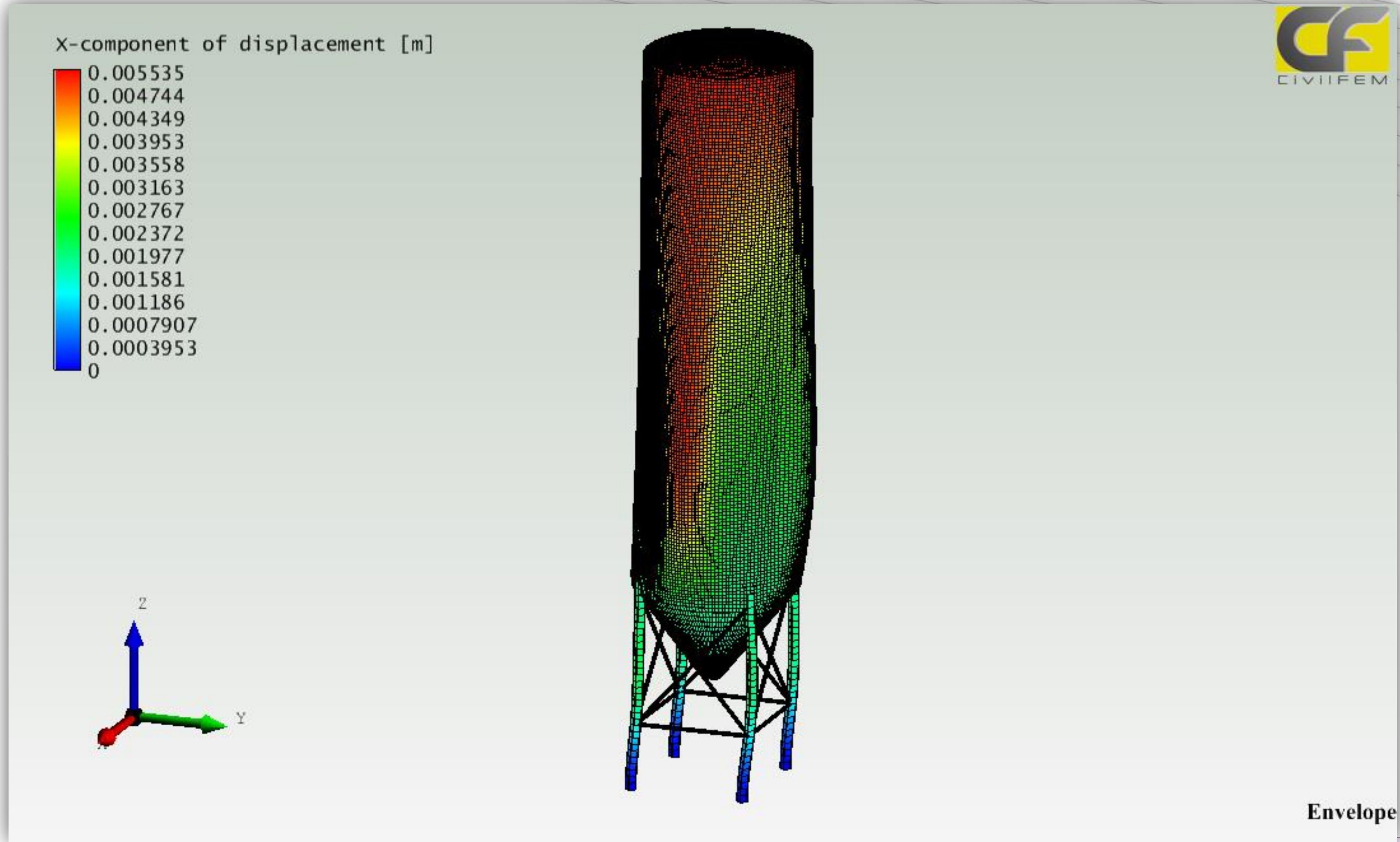


Geometry
Model
Mesh
Loads
Solve
Results
View
Envelope definition
Select results file
Movements
Stresses and Strains
Axial Force
Bending Moment
Torsional Moment
Shear Force
Reactions
Rest of Results
Title
Maximum
Envelope
nformation
List
Synthetic
Envelope
Combinations
Create results
Abrir
silo final tfg
Buscar silo final tfg
Organizar
Nueva carpeta
Favoritos
Descargas
Escritorio
Sitios recientes
Autodesk 360
Bibliotecas
Documentos
Imágenes
Música
Subversion
Vídeos
Equipo
Disco local (C:)
Nombre
Fecha de modifica...
~temp 18/09/2015 9:32
Marc_Run 17/09/2015 12:05
DISCHARGE_SISMO(SRSS).rcf 18/09/2015 9:53
Envelope.rcf 17/09/2015 12:19
FULL STATIC_SISMO(SRSS).rcf 17/09/2015 12:16
SELF WEIGHT_SISMO(SRSS).rcf 17/09/2015 12:16
SW.rcf 17/09/2015 12:16
Wind.rcf 17/09/2015 12:16
Nombre: "DISCHARGE_SISMO(SRSS).rcf" "FULL STAT
Results file (*.rcf)
Abrir
Cancelar
IDWARD
oads
WINDWARD
1.5
nditions
trol

Envelope


Envelope definition
Select results file
DISCHARGE_SISMO(SRSS).rcf
FULL STATIC_SISMO(SRSS).rcf
SELF WEIGHT_SISMO(SRSS).rcf
SW.rcf
Wind.rcf
Movements
Maximum
Stresses and Strains
Maximum
Axial Force
Maximum
Bending Moment
Maximum
Torsional Moment
Maximum
Shear Force
Maximum
Reactions
Maximum
Rest of Results
Maximum
Title
Envelope
Results file Wind.rcf loaded.

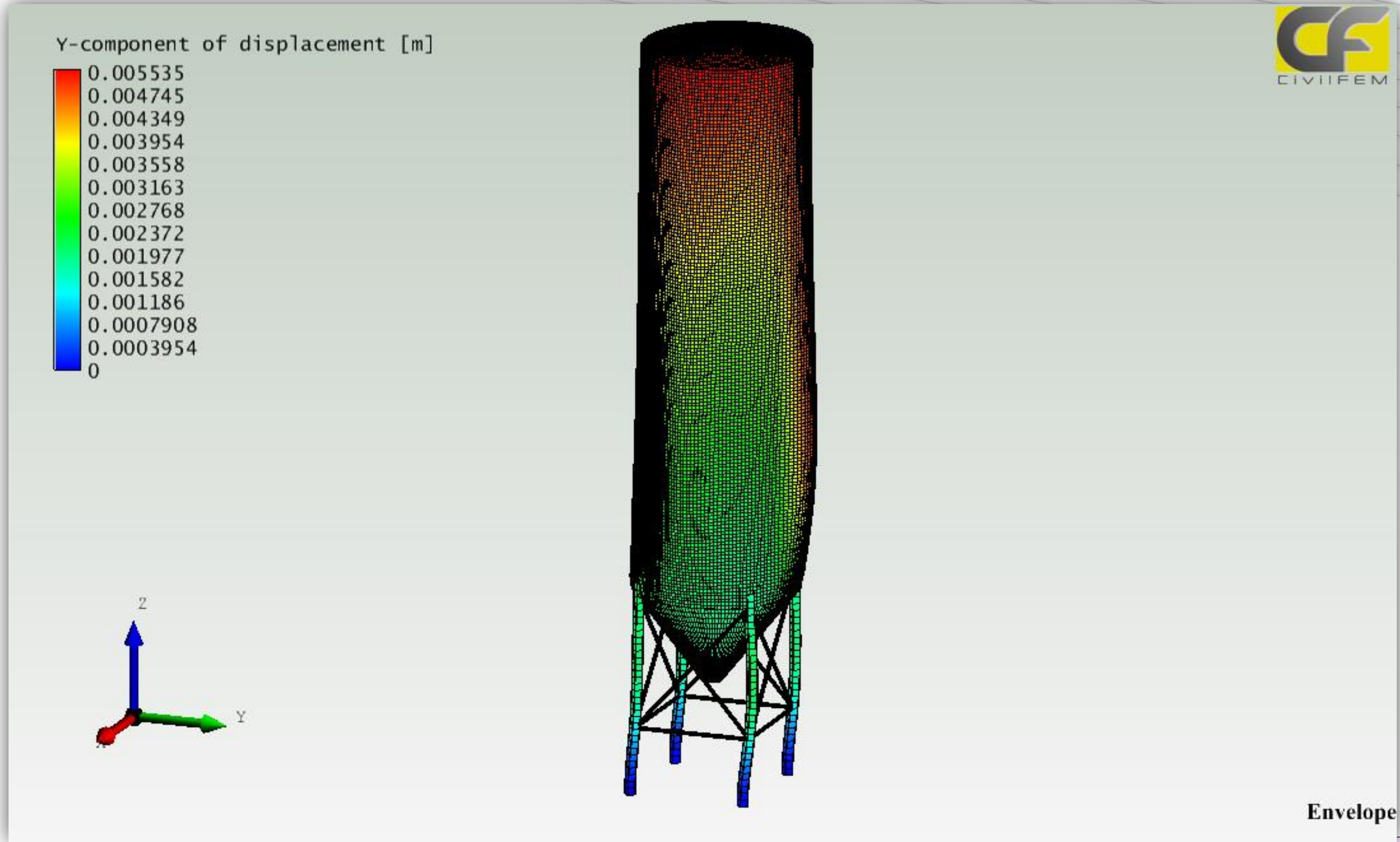
X component displacement



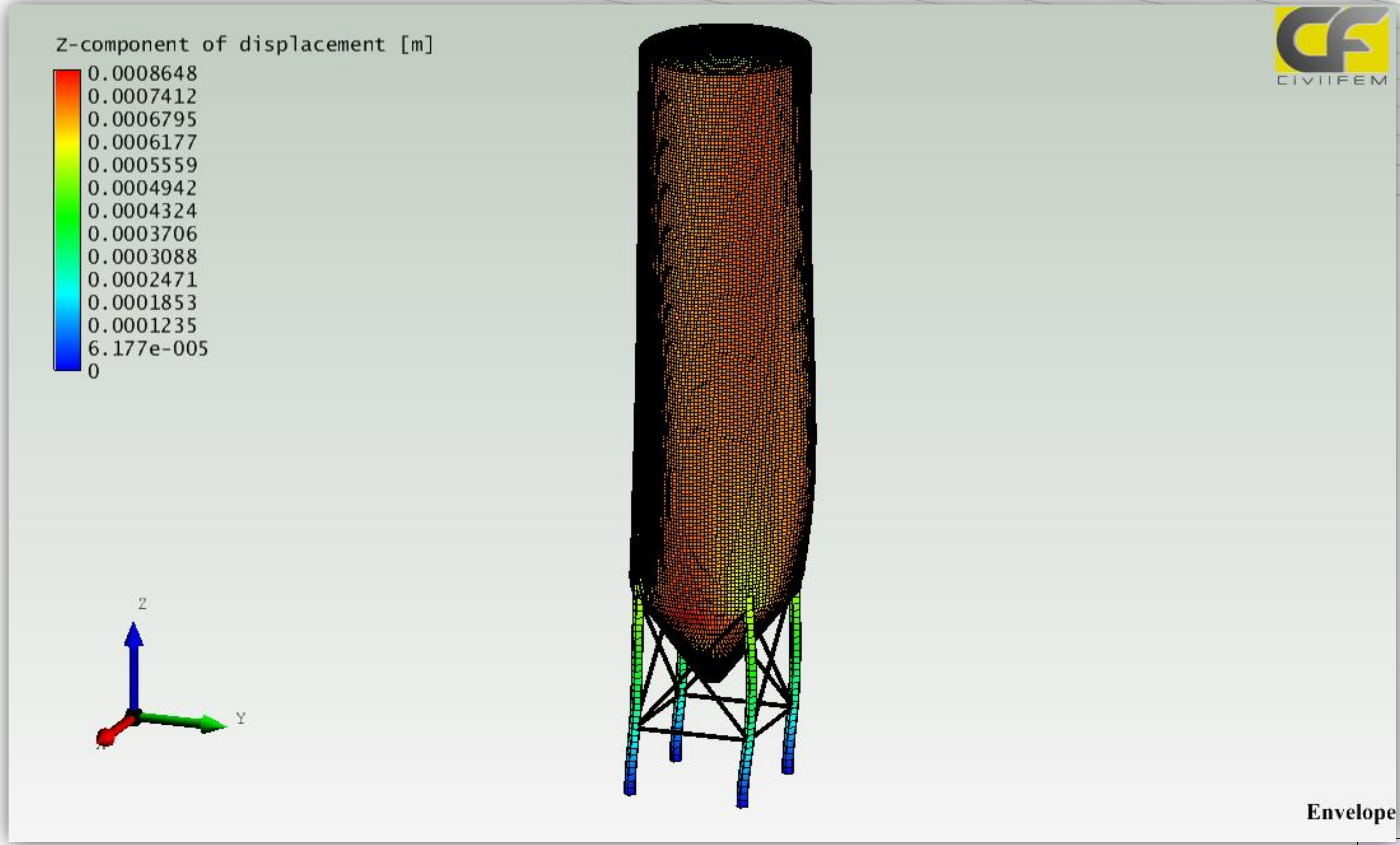
INGECIBER, S.A.

CivilFEM

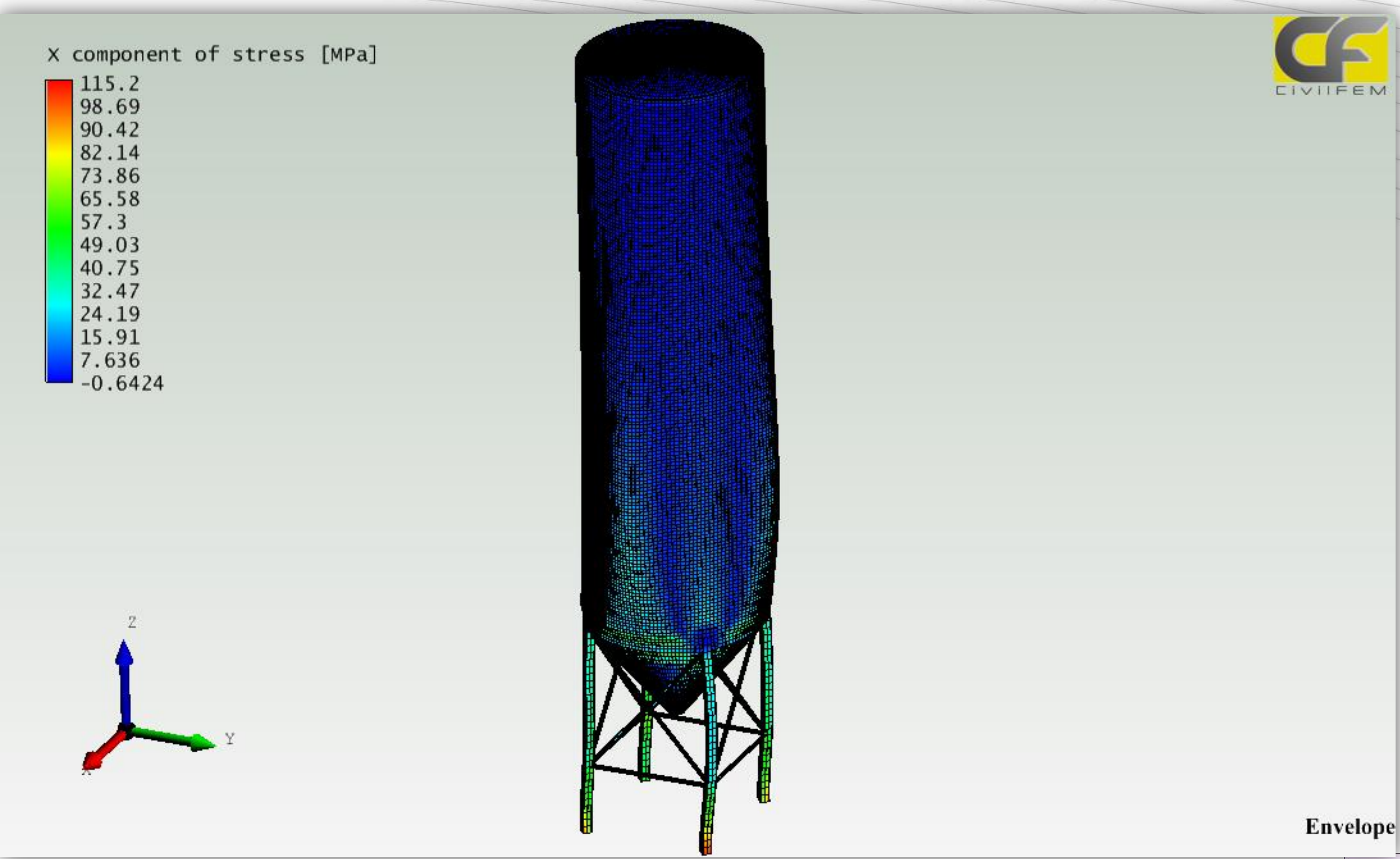
Y component displacement



Z component displacement



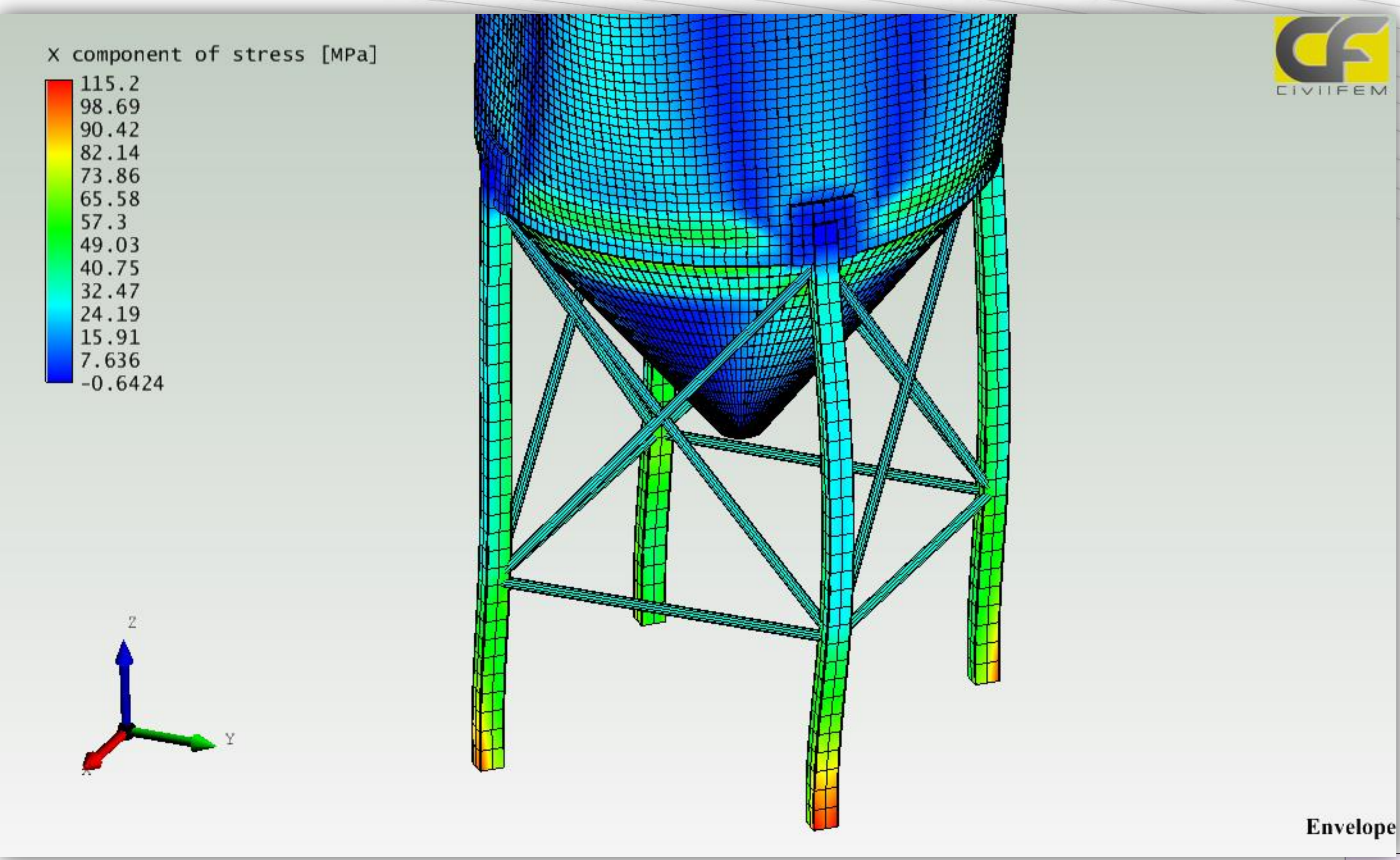
X stress component (I)



INGECIBER, S.A.

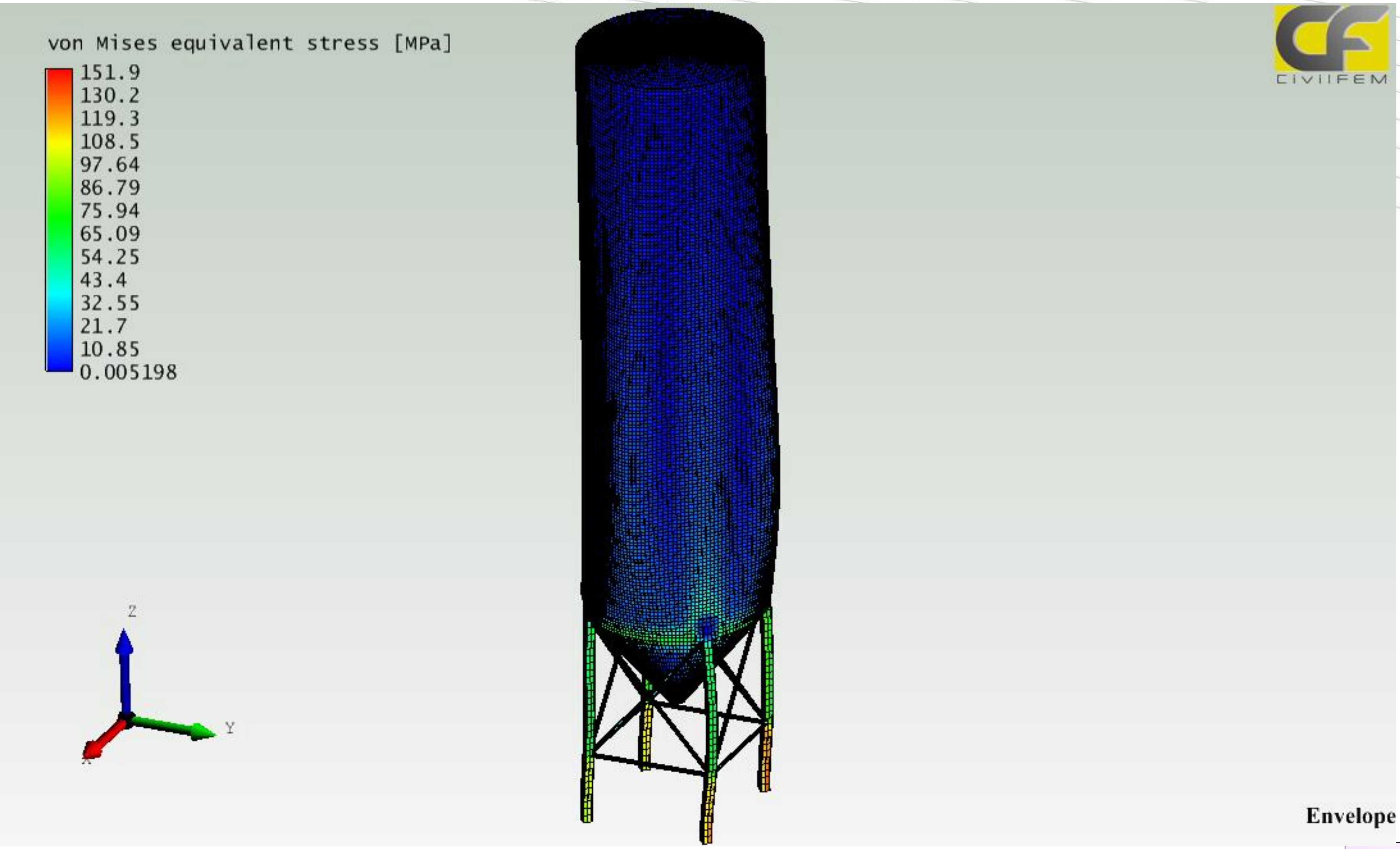
CivilFEM

X stress component (II)



INGECIBER, S.A.

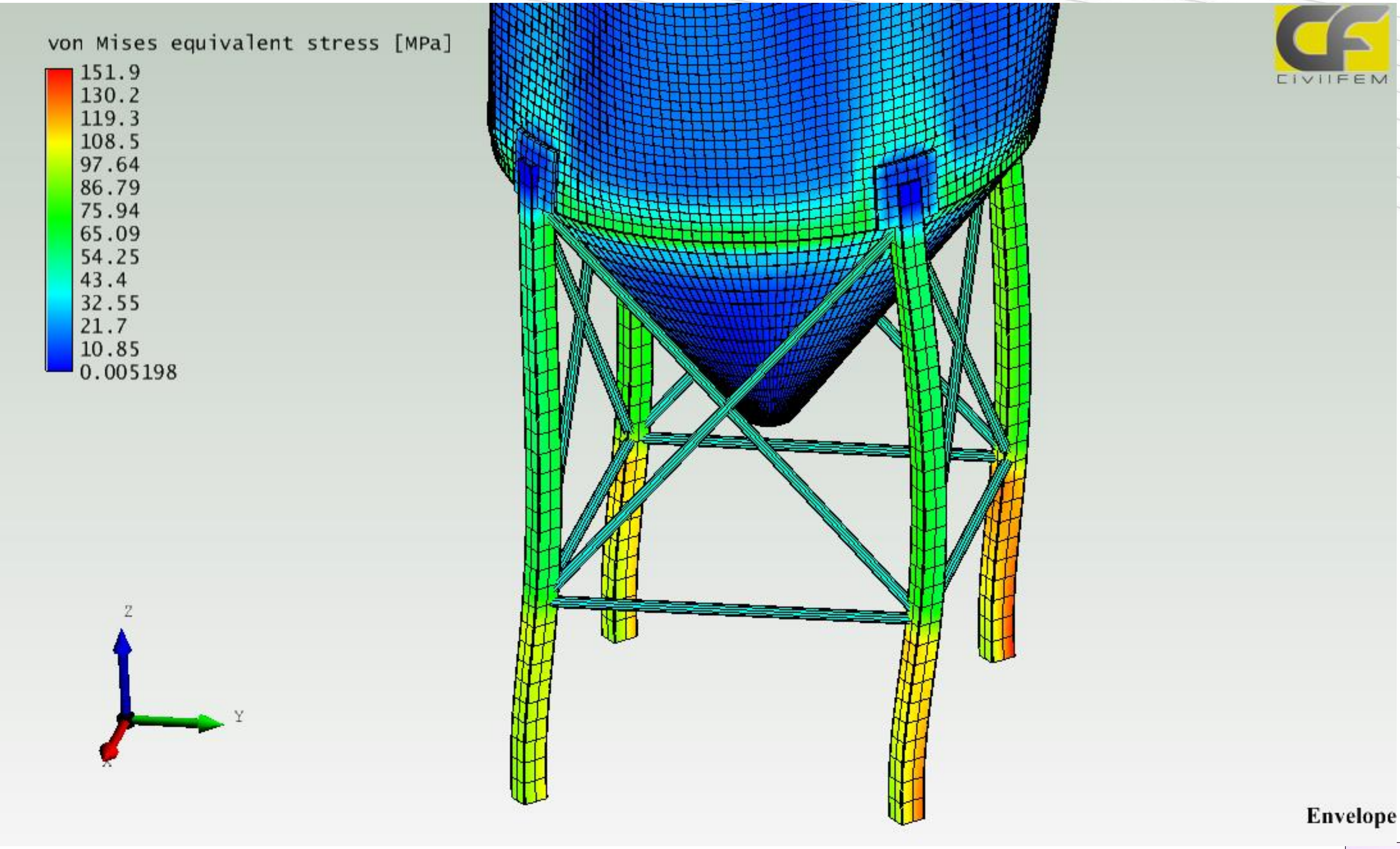
Von Mises equivalent stress (I)



INGECIBER, S.A.

CivilFEM

Von Mises equivalent stress (II)



INGECIBER, S.A.

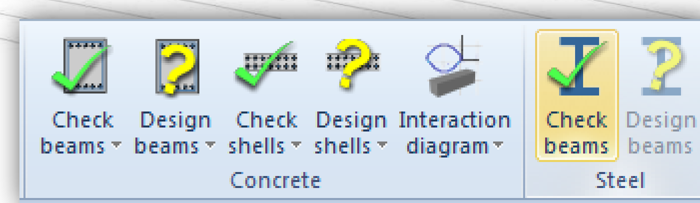
CivilFEM

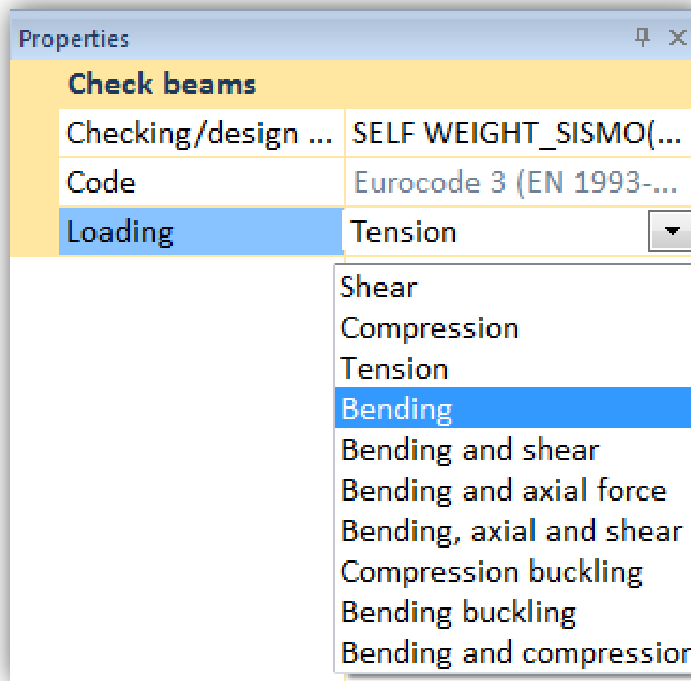
Check

Once a file result is loaded, we can check the structure against a standard. In the Results tab:

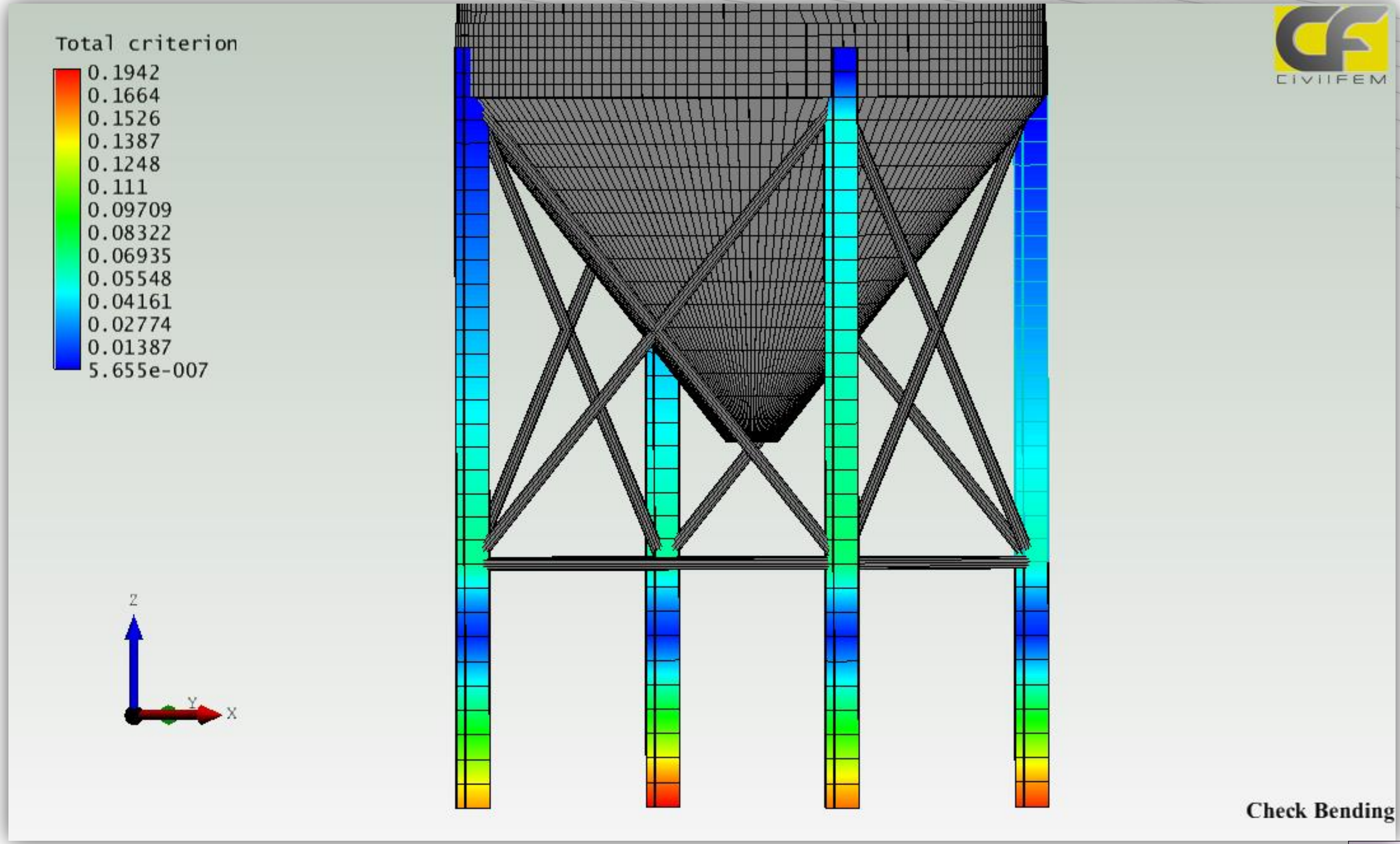
1. Click on **Check beams**.
2. In **Properties** select the **Loading** that we want to check.

In this case we need to check Bending, Compression buckling and Bending, axial and shear force combined. An envelope will be made afterwards.





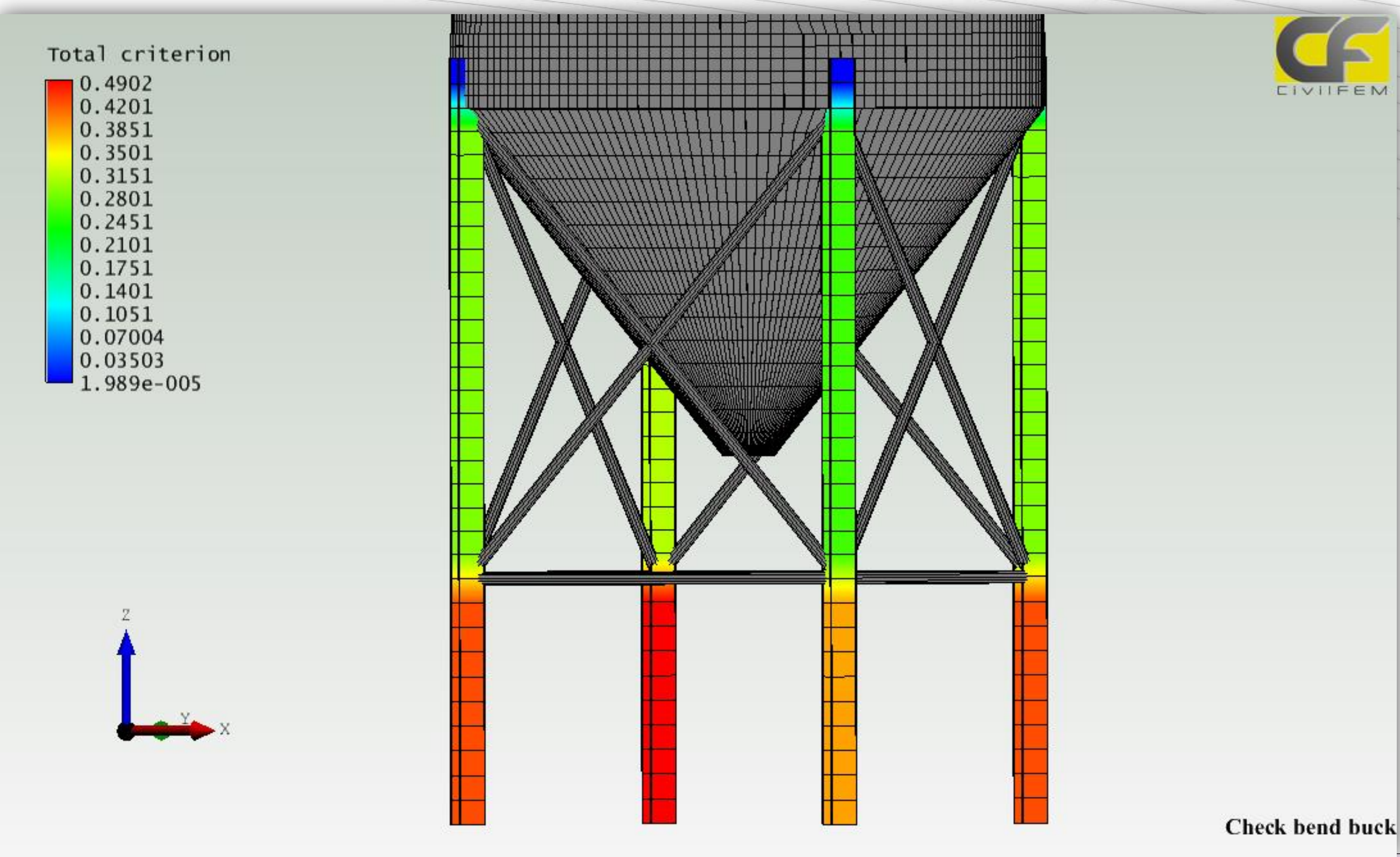
Bending Check



INGECIBER, S.A.

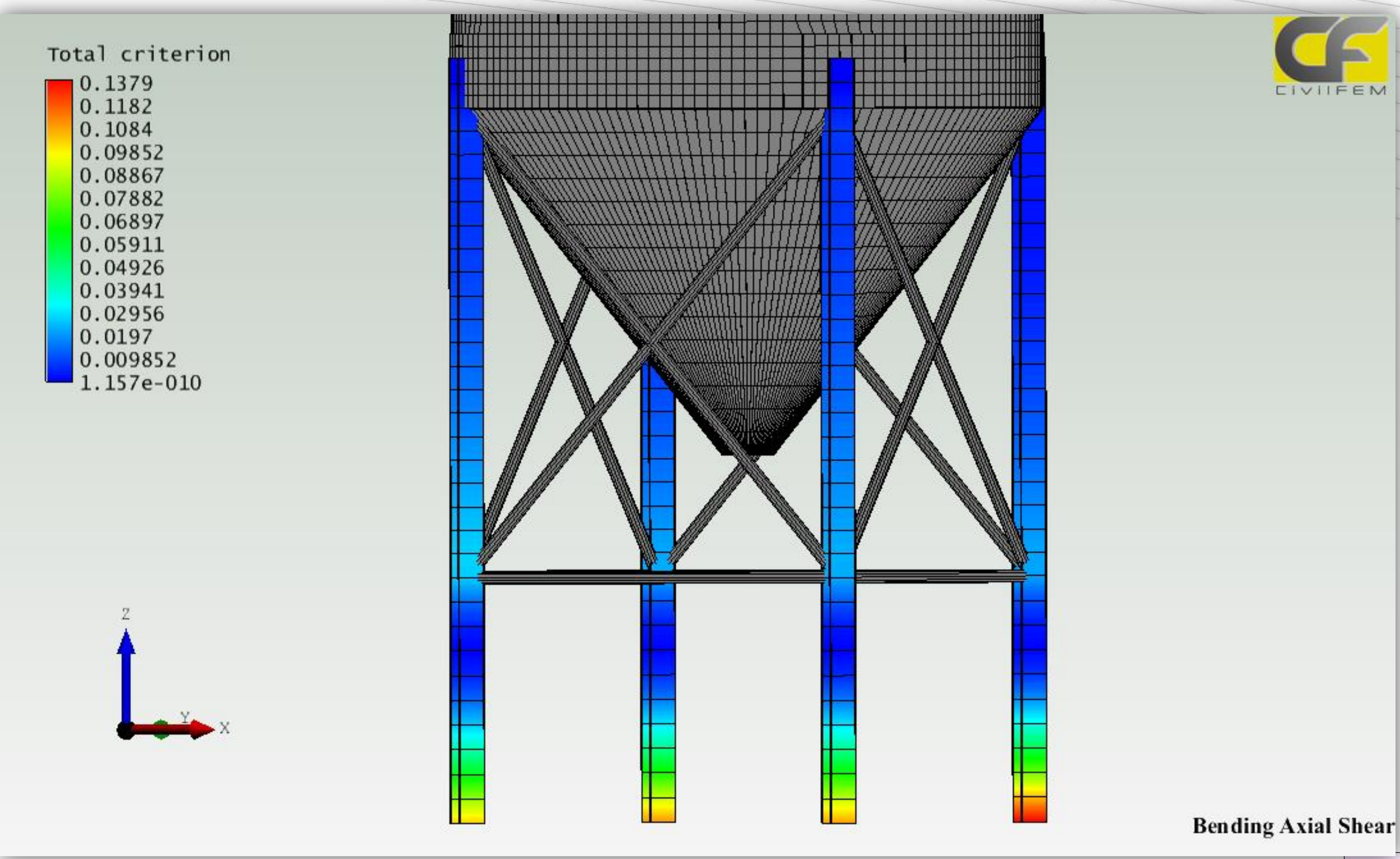
CivilFEM

Bending buckling check



INGECIBER, S.A.

Bending + Axial + Shear check



Conclussions

As we can see in the results, all displacements caused by the seismic load are admissible. In the same way, the generated stresses are lower than the Von Mises comparative stress.

On the other hand, all the loads have a total criterion value lower tan 1,0. This means that the structure is correctly designed.