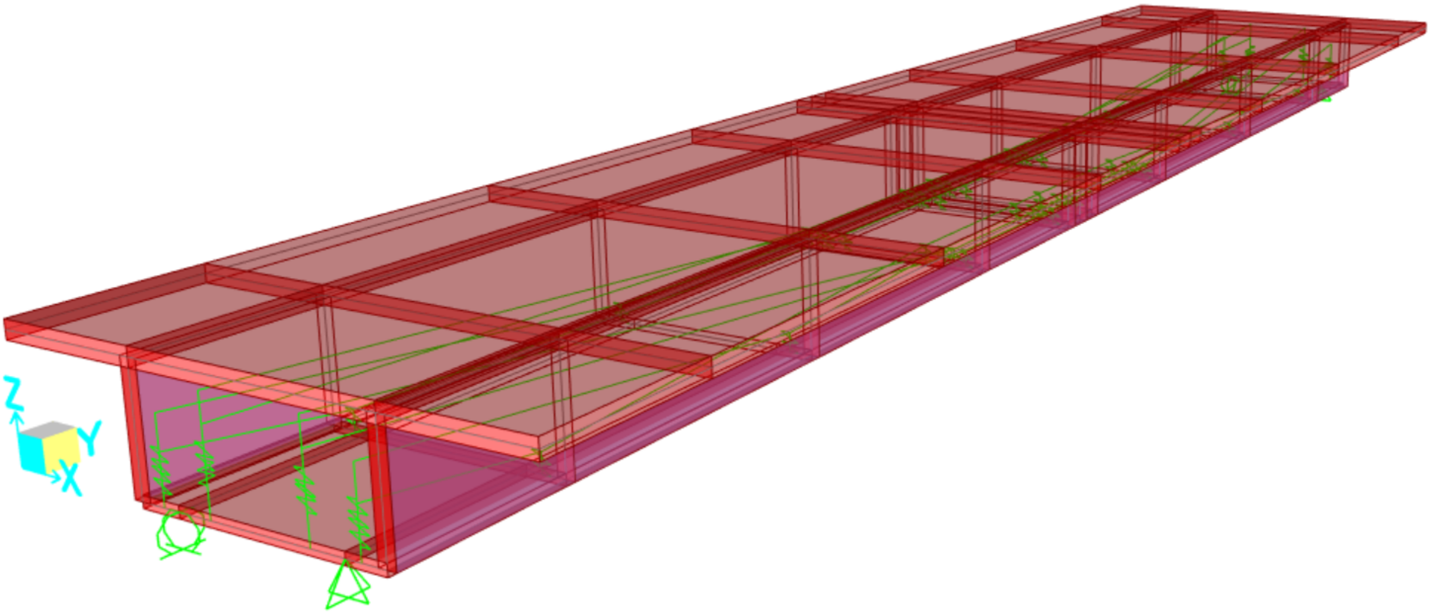
Project on SAP2000

Finite Element course

Anno 2022

Analisi strutturale di un ponte precompresso



Constantin Riff and Bastien Sauvet
Professoressa Addessi

1) Modelization :

We used the software SAP2000 to model four different bridges:

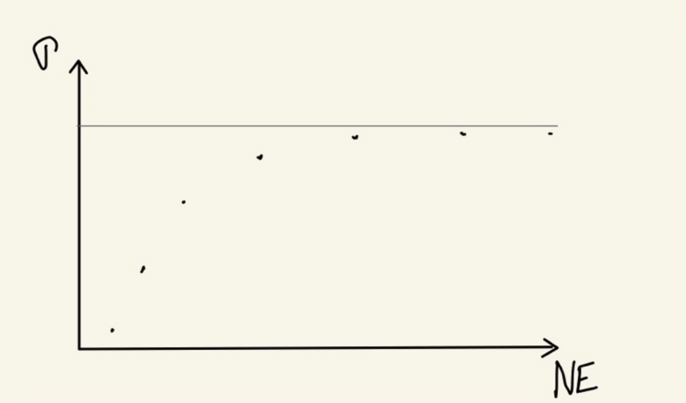
1) One with only pre-stressed loads thanks to tendons, composed only with shell areas for the main structure
2) The same as the first one on which we add two symmetrical distributed loads
3) The same as the first one on which we add two asymmetrical distributed loads
4) Finally, the same shape of the previous ones but composed only with one beam, and pre-stressed with tendons

For the first 3 ones, we used Concrete with γ = 25 kN/m3, E = 30 000 MPa and ν = 0.2. We used steel for the tendon with γ = 78 kN/m3, E = 195 000 MPa and a section for each little cable of 150 mm^2.
In order to model the distributed load, we specially created frame with very low Young Modulus (E = 10MPa) and tiny section (=0,01 m). Thanks to that, these frames are negligeable. We use links to link the different tendons to the structure, at very precise position.

For the last model, we design a single frame thanks to the section designer of SAP2000, with the specific lengths.

2) Results :

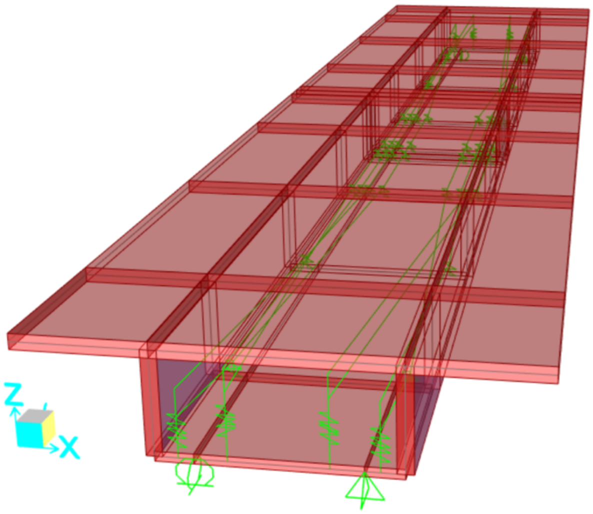
For this models, we were interested specially in some characteristics : the stress *S11* (in the direction of *y*), the stress *S22* but only for the lateral areas (in the -*z* direction), the maximal displacement in the direction u_1 , u_2 and u_3 , and the maximal strain E_{max} . For the stress, we took the values on the top of the areas. We used the command *Show table* to have the result, but also some colormaps in order to know qualitatively where the maximal constraints were. Thanks to the finite element method, we wanted to approximate with accuracy the desired characteristics, so we divided more and more our structure to converge to a value which should be closed to the real one.
Is here the scheme of what we expected.



What we obtained in real life was similar, in some cases we didn't converge as quick as the scheme, and we couldn't divide more our structure to be closer the real value, because it was too tedious for our computer which didn't support it.
We still obtained good results, but sometimes a thinner mesh should have given us a better accuracy.

A) The pre-stressed bridge subjected to its own weight

First of all, we analyzed the bridge subjected only to its own weight, despite the pre-stressed tendons. The initial structure is composed of 80 different area.



First, we divided each area in 4 sub-areas, and then, when the number of areas began to be high, we divided each new area only in two pieces, in the *y* direction, because the mesh should be thin specially along this axis.

Let's look at the result in terms of subdivisions :

1) 1x1 – 80 areas

That's correspond to the initial configuration.
We have for :

- STRESS

S11: +57 MPa and -52 MPa (for the extreme values)
S22: +33 MPa and -10 MPa

- DISPLACEMENT:

U_1 : ± 0,05 m
U_2 : ± 0,037 m
U_3 : - 0,47 m

The U_3 displacement is the one we are the most interested in because it corresponds to the displacement along the *z* axis, the gravity axis.

2) 2x2 – 320 areas

We have for the:

- STRESS:

S11: +123 *MPa* and -77 *MPa* (for the extreme values)

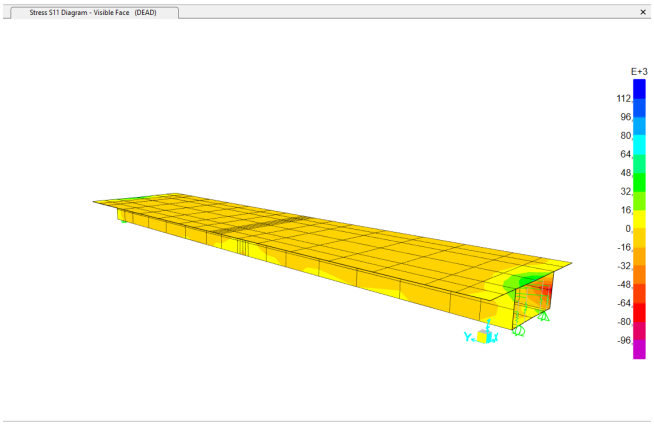
S22: +34 *MPa* and -77 *MPa*

- DISPLACEMENT:

U_1 : ± 0,01 *m*

U_2 : ± 0,03 *m*

U_3 : - 0,45 *m*



1- Colormap of S11 for 2x2

3) 4x4 – 1280 areas

- STRESS:

S11: +196 MPa and – 148 MPa

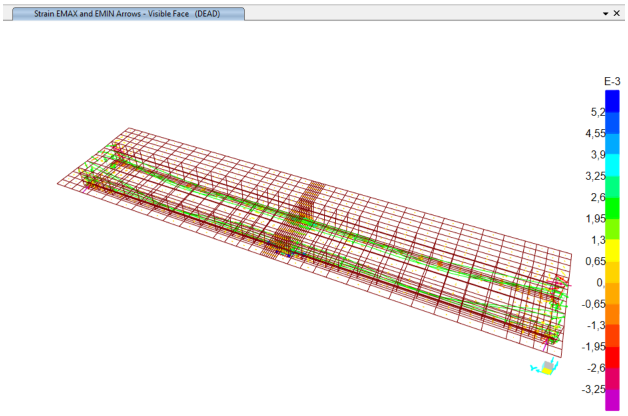
S22: +35 MPa and -109 MPa

- DISPLACEMENT:

U_1 : ± 0,02 m

U_2 : ± 0,038 m

U_3 : -0,46 m



2- Emax (the maximum strain) for 4x4a

4) 8x8 – 5120 areas

- STRESS:

S11: +314 MPa and -272 MPa

S22: +61 MPa and – 164 MPa

- DISPLACEMENT:

U_1 : ± 0,58 m

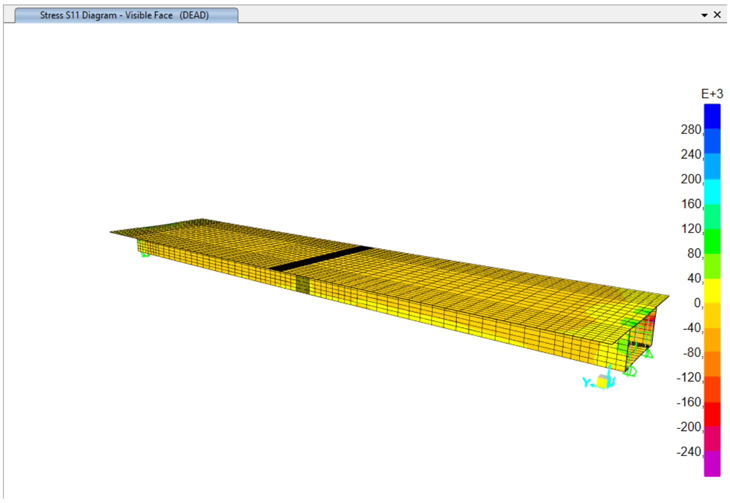
U_2 : ± 0,35 m

U_3 : - 0,46 m

U1 = -X
U2 = -Y

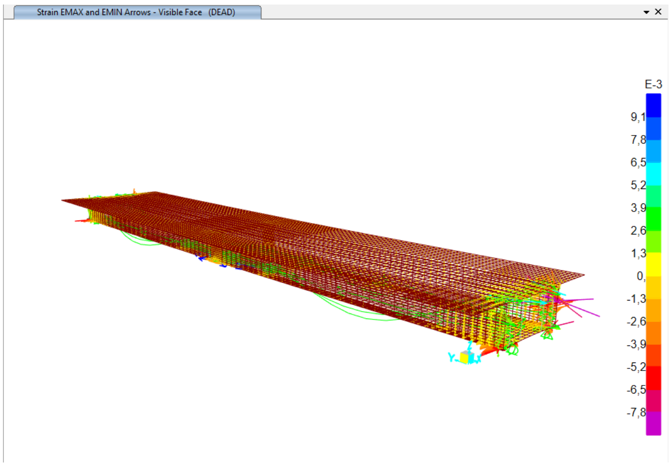
Nouveau :
U1 = 0,2
U2 = 0,035
U3 = -0,46

At this point, we divided only along the *y* direction.



3- Colormap of S11 for the 8x8

5) 8x16 – 10240 areas



4 - Colormap of Emax for the 8x16

- STRESS:

S11: +444 MPa and – 297 MPa
S22: +136 MPa and -173 Mpa

- DISPLACEMENT:

U_1 : ± 0,15 m
U_2 : ± 0,36 m
U_3 : -0,46 m

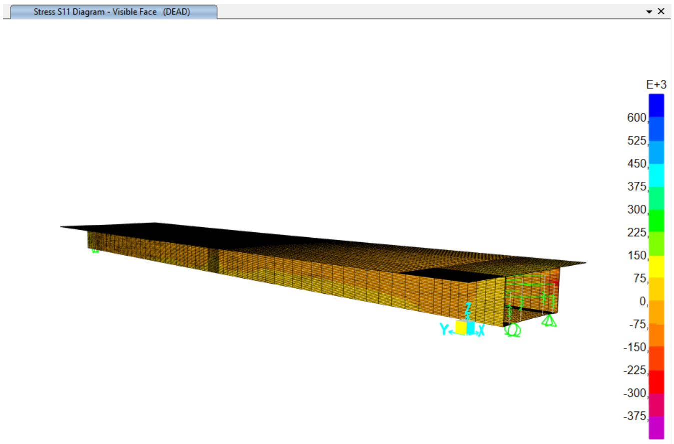
6) 8x32 – 20480 areas

- STRESS:

S11: +613 MPa and – 336 MPa
S22: +206 MPa and – 173 MPa

- DISPLACEMENT:

U_1 : ± 0,41 m
U_2 : ± 0,05 m
U_3 : - 0,46 m



5 - Colormap of S11 for the 8x32

7) 8x64 – 40960 areas

- STRESS:

S11: +770 MPa and -472 MPa
S22: +268 MPa and -216 MPa

- DISPLACEMENT:

U_1 : ± 0,75
U_2 : ± 0,08
U_3 : -0,47

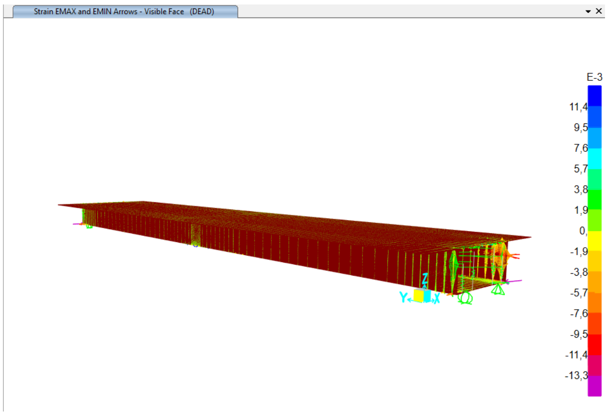
8) 8x128 – 81920 areas

- STRESS:

S11: +855 MPa and -755 Mpa
S22: +307 Mpa and -245 MPa

- DISPLACEMENT:

U_1 : ± 1 m
U_2 : ± 0,09 m
U_3 : -0,47 m



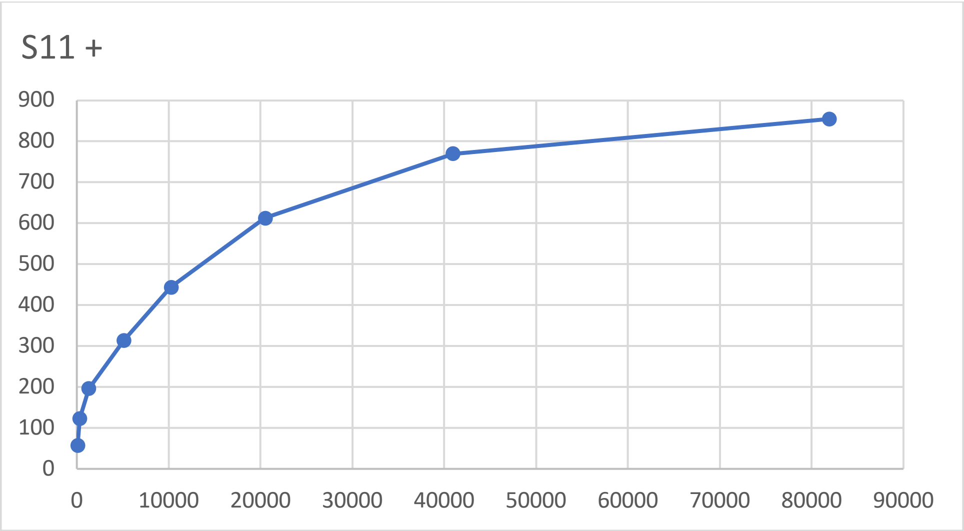
6 - Colormap of Emax for the 8x64

We couldn't reach a thinner mesh because of the computing power.

Let's summarize of the values in a table, and look at the convergence :

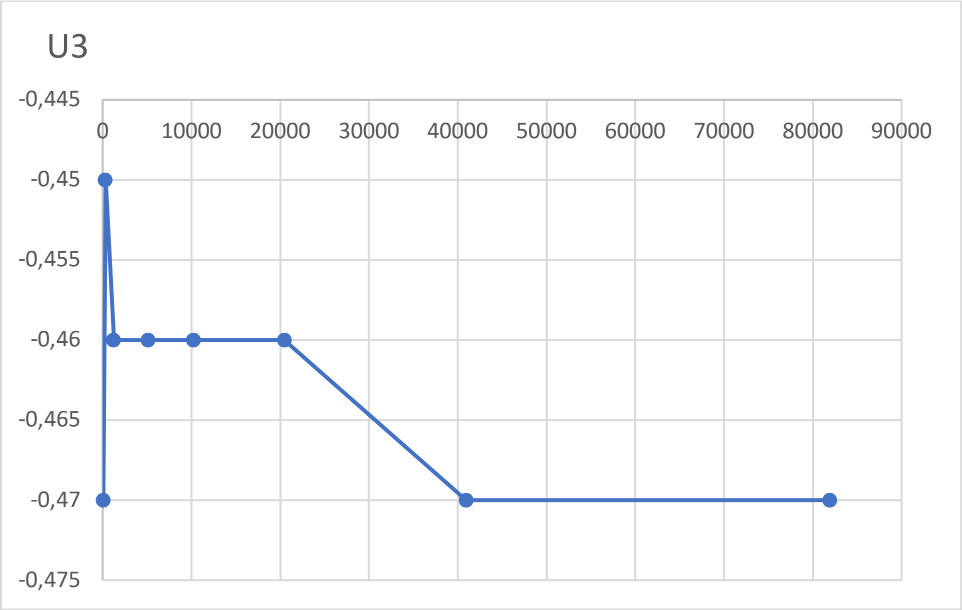
Self-weight	(MPa)				m		
Number of Areas	S11 +	S11 -	S22+	S22-	U1	U2	U3
80	57	-52	33	-10	0,05	0,037	-0,47
320	123	-77	34	-77	0,01	0,03	-0,45
1280	196	-148	35	-109	0,02	0,038	-0,46
5120	314	-272	61	-164	0,58	0,35	-0,46
10240	444	-297	136	-173	0,15	0,36	-0,46
20480	613	-336	206	-173	0,41	0,05	-0,46
40960	770	-472	268	-216	0,75	0,08	-0,47
81920	855	-755	307	-245	1	0,09	-0,47

We did some graphics in order to analyze the convergence of our solution :



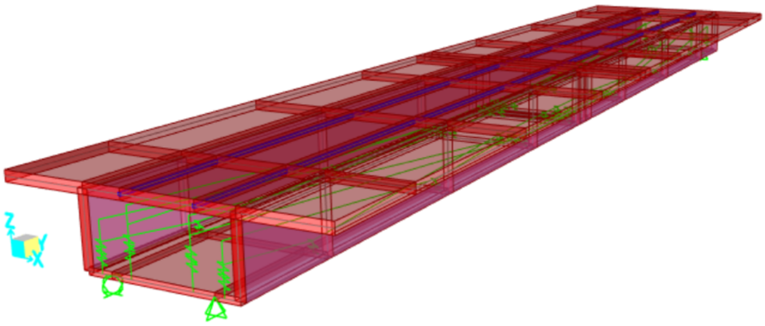
On this one, we can see that the value of the stress in the *y* direction converge slowly to a value around 870 MPa. We observe the same fashion for the other stress measurements.

For the displacements, for U_1 and U_2, we can't see a general fashion, the value is always small but there is variation between the different meshes. For U_3, we obtain a very precise value since the beginning, and this value doesn't change a lot with the different meshes.



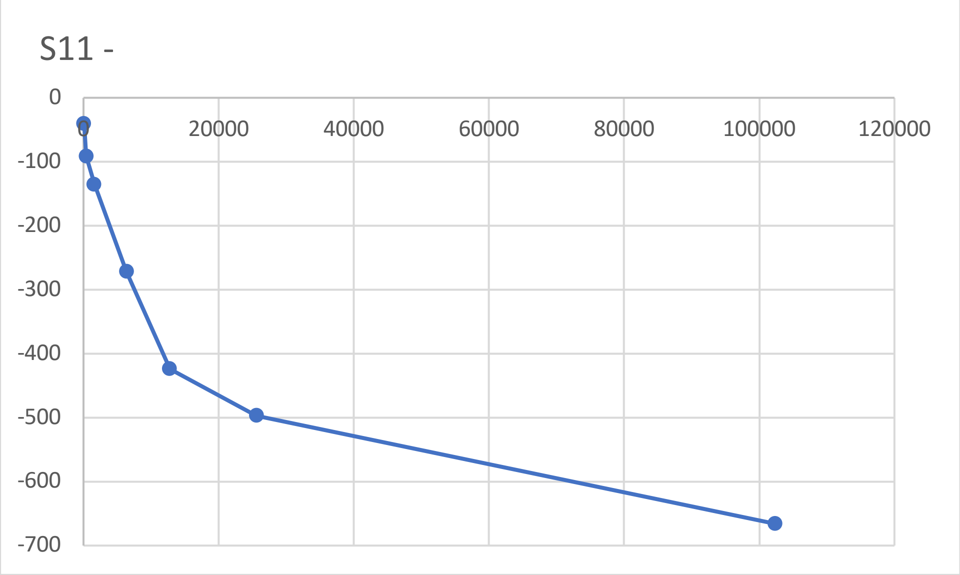
B) <u>The pre-stressed bridge under a symmetrical distributed load</u>

In this model, the bridge was the same as the previous configuration, but it was subject to a symmetrical linear distributed load, which were applied on the blue frames on the next picture. The value of the load was *35 KN/m.*
The initial configuration is composed of 100 areas.



In order to have clearer results, we summarize all the results in a table.

Symetric load	(Mpa)				(m)		
Number of areas	S11 +	S11 -	S22+	S22-	U1	U2	U3
100	58	-40	20	-19	0,02	0,034	-0,455
400	127	-91	40	-50	0,004	0,005	-0,062
1600	198	-135	69	-113	0,005	0,005	-0,068
6400	316	-272	90	-157	0,005	0,006	-0,07
12800	343	-424	125	-370	0,006	0,006	-0,071
25600	515	-497	133	-393	0,006	0,006	-0,07
102400	866	-666	300	-357	0,007	0,007	-0,071



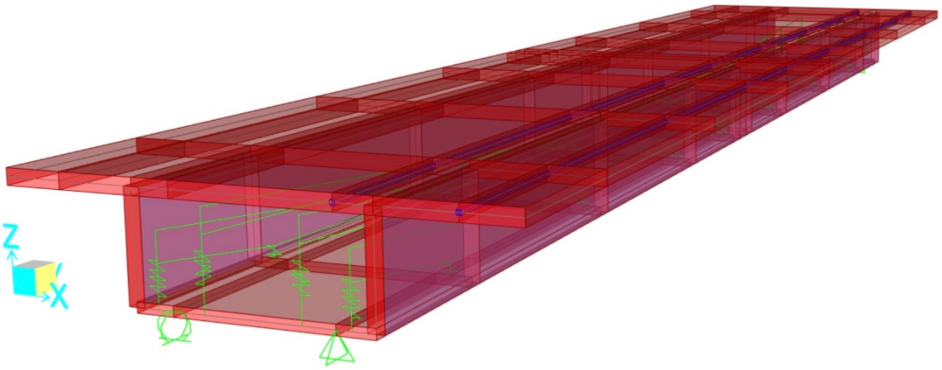
We can see on this graph that we have also a convergence of the value of S11 (for the negative value in this example), but we can notice that we could have a better approximation with a thinner mesh.
The values of U_1 and U_2 don't vary a lot, we can notice that for both, with the mesh with 100 areas, the value is much bigger that the next we obtain.

For U_3, we see that we have also a high value for the first mesh, and then the value turns around -0,07.

C) The pre-stressed bridge under an asymmetrical distributed load

In the model, the bridge was as the first configuration with two asymmetrical linear distributed loads which were applied on the blue frames. The value of the loads were still *35 KN/m.*

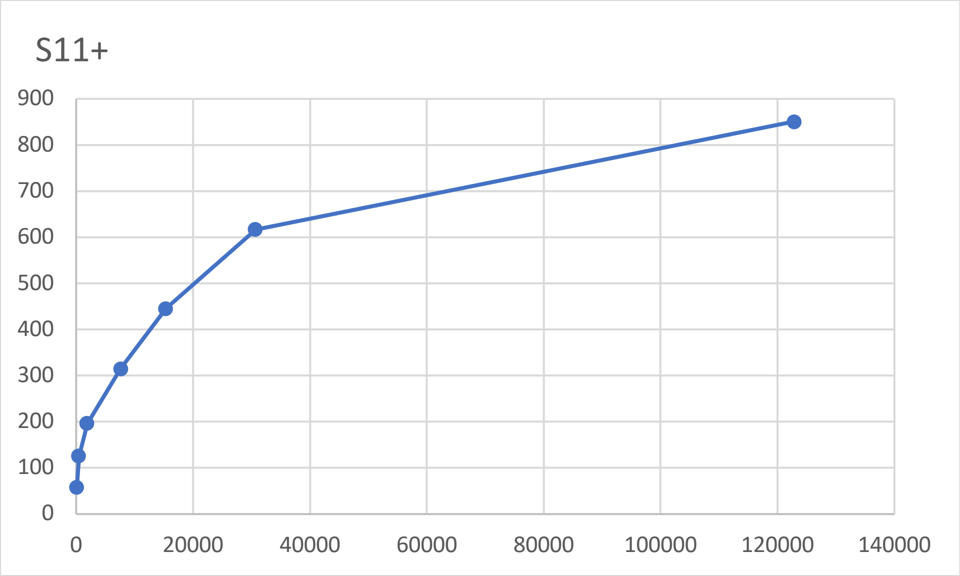
The initial configuration is composed of 120 areas.



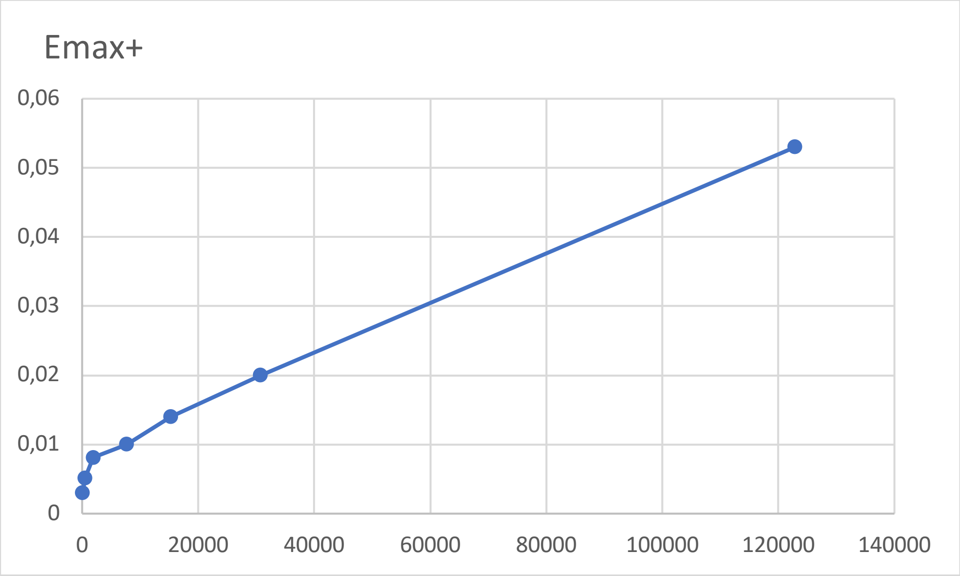
Here are the results :

Asymmetric load	(MPa)		Ø		m			
Number of areas	S11+	S11-	Emax+	Emax-	U1+	U1-	U2	U3
120	58	-37	0,002989	-0,000415	0,025	-0,025	0,033	-0,45
480	126	-104	0,00509	-0,0014	0,035	-0,017	0,038	-0,49
1920	197	-188	0,0081	-0,0011	0,038	-0,005	0,033	-0,47
7680	315	-274	0,01	-0,0019	0,08	-0,04	0,031	-0,469
15360	446	-298	0,014	-0,0019	0,17	-0,12	0,038	-0,471
30720	617	-337	0,02	-0,0039	0,46	-0,39	0,06	-0,479
122880	851	-741	0,053	-0,007	1,33	-1,24	0,13	-0,49

We can see that all the stress measurements converge with the increasing of number of areas, which means the thinner our mesh is. For example, we have below, the graph of S11 with respect to the number of areas.

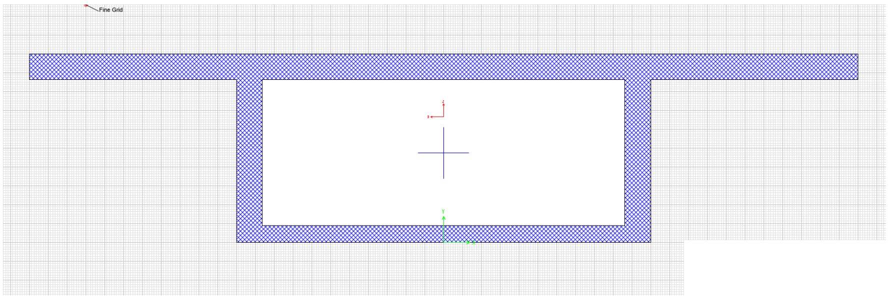


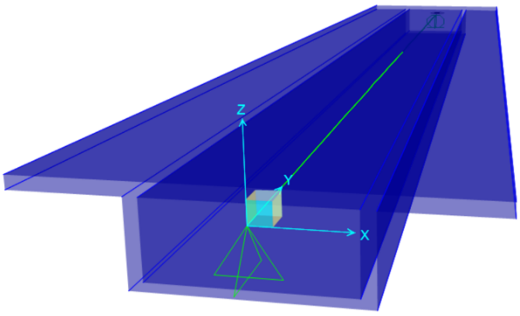
The deformation grows linearly with the number of areas as the next graph shows us.



D) The one-beam model bridge

The last model in composed for the structure of only one beam whose the section is the same as the previous models. We applied the same tendons, but it was only along one direction while in the previous models, the tendons were also along the *x* axis and the *z* axis. For the restraints, a beam need only two supports, so we set a pin at one of the extremities, and a roller at the other one.





We obtain for this model that : the maximum axial stress is 1170 MPa, the maximum displacement along the *y* axis is : U_2= -0,079 m and the maximum displacement along *z* is U_3= -0,0865 m.

Of course, because we used a continuous beam, if we divide this beam in many pieces, the results will be the same.

3) Comparisons between the different models

We will first compare the 3 shell-area's models.

We see first that the difference for the maximum stress S11 is not very high between the models, which is not very intuitive. We could expect that this value would be higher for the structures subjected to the distributed loads.

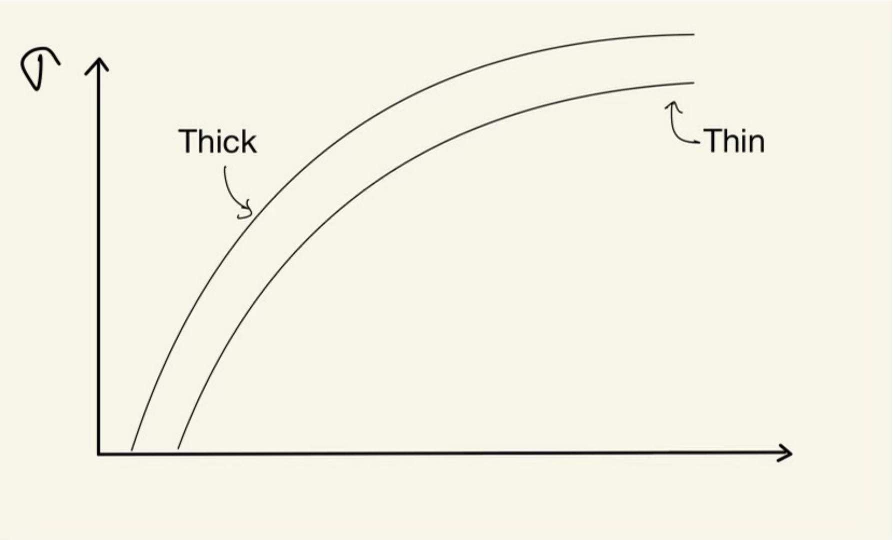
We could suppose that it is thanks to the tendons which pre-stressed the structure and absorb a part of the distributed loads.

We can formulate the same behavior for the other stress components.

About the vertical displacements, this former is the higher for the model with the symmetrical loads. We can explain that because, on the first model, there is no external load so the deformation will be smaller, and for the third model, the loads are asymmetrical so there is more shear stress, and so the axial stress is smaller than in the symmetrical model, and so we expect less displacement along *z*.

Moreover, if we compare the three first models and the one-beam model, we can see on the last one that the axial stress is higher than the S11 stress of the shell models. We can explain that difference because the beam will be subject to pure bending and so all the different forces will be reflected in the value of the axial stress.
Moreover, the displacement of the beam is smaller than the displacement of the previous models, we can explain this difference because the beam is a single element, and so it is less malleable than a multi-element structure.
We can see the fourth model as a coarse approximation of the area shell models, the obtained values are not far away from the ones of the previous models, but there are still less precise.

We can precise that we decided to use thick shells rather than thin shell, because it includes the transverse shear deformation in plate-bending behavior (like the Mindlin/Reissner formulation) while the thin shell follows the Kirchhoff formulation.
This difference will be not negligeable anymore for elongated elements where the shear deformation tends to be important, which can happen in our models.
Thick shells tend to be stiffer than thin shells, so we could see a difference between a thin and a thick model, as showed in this scheme.



4) Conclusion

Thanks to this project, we first learned how to use the software SAP2000 which was totally unknown for us before. For the first time we used computer in order to model and compute loads in a usual structure, and used our theorical knowledge for an applicate case.
We could compare different models for the same structure, notice the differences, the pros and cons of each and realize the importance of the different parameters. We also took note of the benefits of the computational analysis, in particular the finite element method.
This project allows us to know better the civil engineering’s fields, that is very important for the choice of our master for the next years.