1 Triangular elements

In this section, a system composed of a triangular element, subjected to both concentrated and distributed loads, is analyzed using FEM methodology. Essentially, the main steps to solve the problem are:

- Define a displacement function for both displacements u and v. Since the expressions are equal, except for the constants, it is possible to analyze just the case for the u displacement for the evaluation of the shape functions. In the analyzed case they will be

  $u = c_1 + c_2x + c_3y + c_4xy$

  $v = c_5 + c_6x + c_7y + c_8xy$

- Calculate the shape functions by substituting the local coordinates into the expression to find the u-displacements for all the nodes. As a result, a system of 4 equations will be achieved and solving it will lead to the calculation of the constants.

- Derivation of the expression for the strain-displacement matrix by calculating all the components of the strain vector as a function of the displacements

- Determination of the Force Vector by splitting up the two loading cases, determine the effect of both and sum up the two effects.

In the following the whole analytical process is illustrated.

1.1 Shape functions - Question 1a

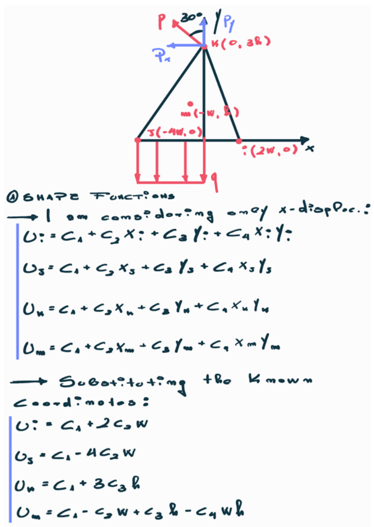

Figure 1: Introduction to the problem

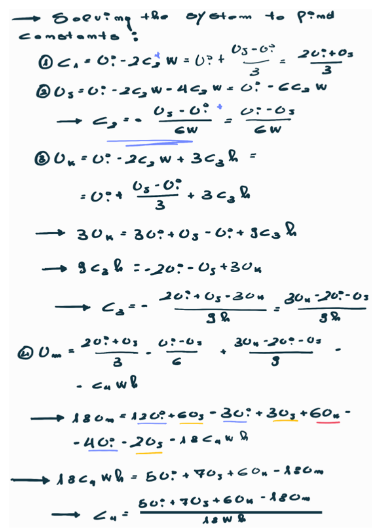

Figure 2: Determination of constants

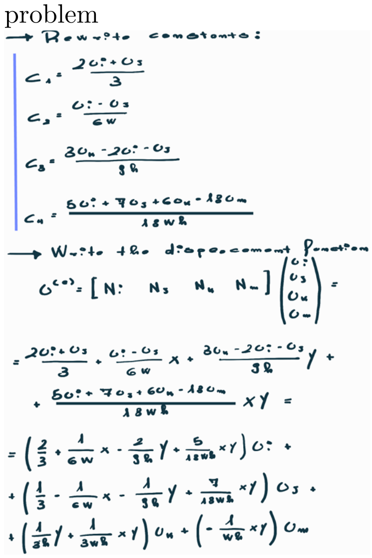

Figure 3: Re-writing of the u-displacement function

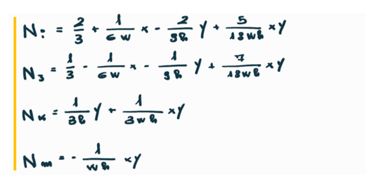

Figure 4: Final shape functions

1.2 Strain-displacement matrix B - Question 1b

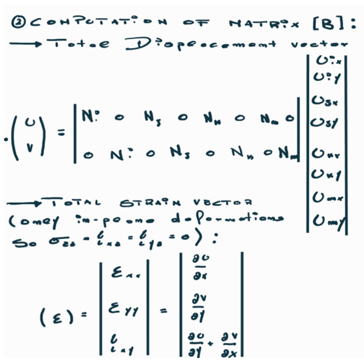

Figure 5: Expressions for displacement and strain vector

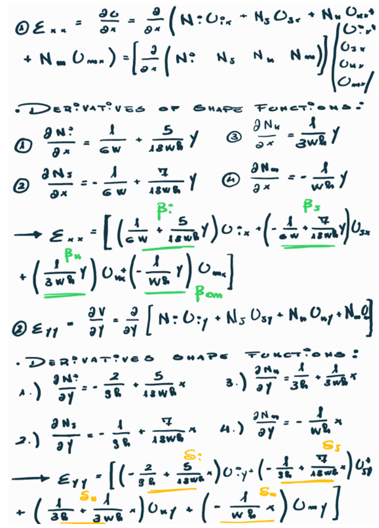

Figure 6: Estimation of the derivatives of the shape functions and strain components

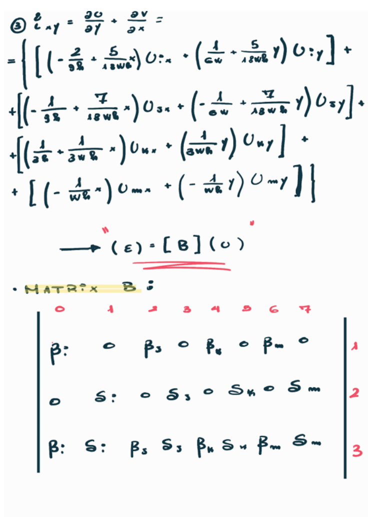

Figure 7: Writing of the strain-displacement matrix **B**

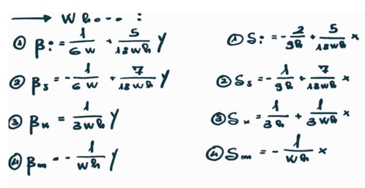

Figure 8: Coefficients of the matrix

1.3 Difficulties of integration and possible solution - Question 1c

In order to compute the global stiffness matrix, it would be necessary to evaluate the stress-strain matrix **D** and perform the volume integration $\int_V [B]^T[D][B]dV$, since the terms of the stiffness matrix will be obtained by deriving the strain energy. However, the integration may not be so easy in this case because the terms in the strain-displacements matrix **B** depend on both x and y, so the integration would require to evaluate how these components vary across the volume domain V. In the case discussed in class, instead, the terms of both matrix **B** and **D** were constants and that contributed to a much easier integration.

Usually, these kind of operations are made in FEM analysis using **Gaussian quadrature**, which is a type of numerical integration. Basically, the main step is to approximate the integral with the summation of a weight multiplied by a function value at location x_i.

$$\int_a^b f(x)dx \cong \sum_{i=1}^{n} w_i f_i(x_i)$$

In this way, it is possible to substitute the integrand with a function that depends on a variable λ that goes from -1 to 1. As a consequence, also the extremes of integration will be equal to -1 and 1, so it is no longer required to scale the integration domain every time there are different bounds. Hence, the integral to approximate with the summation will be:

$$\int_{-1}^{1} f(\lambda)d\lambda$$

Computing the weights and the functions will depend by the number of sampling points, but in general one can assume polynomial functions, such as the **Legendre polynomials**, to extrapolate the necessary equations for the calculation of the coefficients.

1.4 Computation of Force Vector - Question 1d

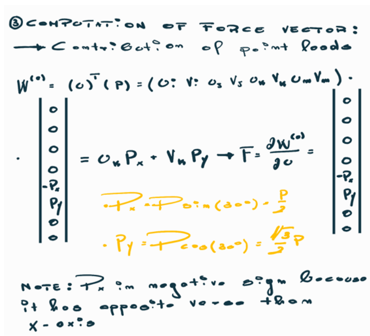

Figure 9: Estimation of the concentrated load contribution

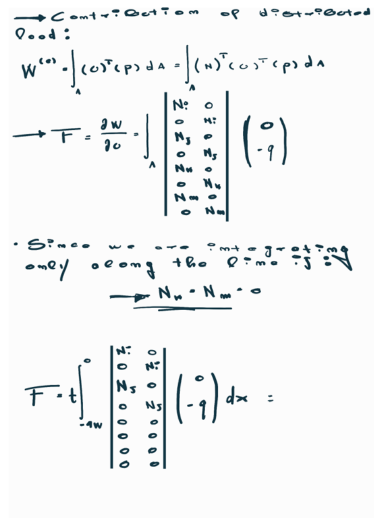

Figure 10: Evaluation of the distributed load contribution

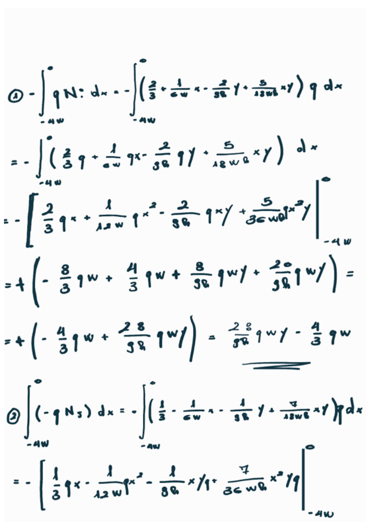

Figure 11: Resolution of the integral

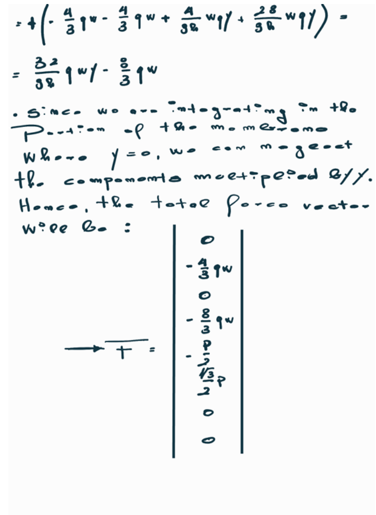

Figure 12: Determination of the force vector

2 Panel under tension

2.1 Real-life application - Question 2a

A clamped panel subjected to the action of many small weights attached underneath it will be tested. The goal of this section is to analyze the test and optimize it by using ABAQUS. As a first step, it is necessary to think about which real-life applications the test could simulate; in other words, why a panel clamped at the top with a loading beneath it should be tested?

Take for example, the **indoor digital billboard at the Tokyo train station** showed in the figure below. It can be seen that the billboard is clamped at its top portion and the encastre may also involve a little portion of it, giving an accurate simulation of the tested clamp distributed across the top 50 mm of the panel. Furthermore, there are other smaller spotlights attached to the bottom of the digital billboard, which embody the small weights attached to the panel in the test.

Figure 13: Digital billboard at the Tokio Train Station

So actually this test can simulate effectively real-word situations. However, assumptions have to be made in order to set-up the test in a much easier way, particularly the assumption of **uniform clamp at the top portion of the panel**. From the authentic example considered, it can be inferred that the clamp system may not be uniform across the top size of the billboard.

Essentially, instead of having one continuous clamp along the whole top edge, the system is composed of several attachments that connect the billboard to the ceiling and they are positioned to the corners and center of it. This because those are the areas where structural integrity is most needed. Hence, the connection in these kinds of real-word application is likely to be non-uniform; however, in order to make the test set-up easier it is possible to assume the constraint system as uniform, but obviously it will not be a perfect representation of reality, because it is assumed that there are no stresses at the top edge when small displacements and stresses may still be present.

2.2 Idealisation of test and set-up in a Finite Element Analysis - Question 2B

Firstly, the essential steps to idealise the test according to Finite Element Analysis are listed below. The **main two assumptions** are that, as previously said, no stresses at the clamped portion have been assumed when in reality there may be small deformations and displacements; the other one is that the total action of the small weights is simulated as a distributed load when actually they simply represent small concentrated forces.

- Definition of the problem geometry
- Specification of Materials properties
- Mesh of the panel
- Application of Boundary conditions
- Implementation of loading conditions

2.2.1 Problem Geometry

Model the panel effectively with respect to panel's dimensions (Height, width and thickness). Also the constrained top edge of the panel should be modeled accordingly to its sizes. Moreover, an element type has to be chosen in order to proceed with the simulation of the problem: for the model elaborated a solid shell element is chosen with a thickness of 2 mm.

2.2.2 Specification of Material Properties

Definition of Young's Modulus (E = 70 GPa and $\upsilon = 0.3$ in this particular case) and Poisson coefficient of the material. In this case, the assumption considered is that the behavior of the material is limited to the linearly elastic portion, which means that the relationship between stress and strain is linear.

2.2.3 Mesh of the Panel

The third step is to mesh the panel using a specific element type for the panel. A Linear Quadrilateral Element (S4R) has been applied and 20x20 quad meshing elements have been used into the ABAQUS model.

2.2.4 Application of Boundary conditions

The boundary condition of zero displacement in every direction of the coordinate system has to be taken into account for the top edge.

2.2.5 Implementation of loading condition

Loading system is finally simulated by using both concentrated or distributed loads. In this case, loading system is reproduced using a Shell Edge Load applied beneath the panel.

2.3 Discussion of results obtained from the ABAQUS model - Question 2C

After implementing the steps explained above, the following results are achievable

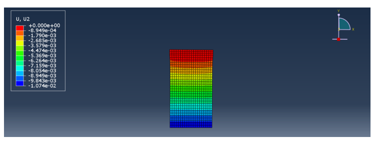

Figure 14: Displacements along y-axis of the original panel

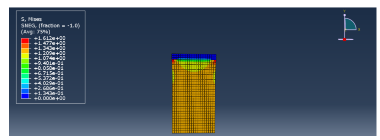

Figure 15: Mises Stress across the original panel

- **Displacements** - As one may expect, displacements along the y-axis will increase from the constraint to the bottom due to the action of loads, hence the model is consistent with the results despite the assumptions. As a result of them however, there will be no displacements at the top but it is important to understand that in reality there could still be small displacements at the constraint area.

- **Stresses** - The stress distribution is symmetric throughout the whole section of the panel, and stress concentrations at the top corners just under the constraint are present, obviously for geometrical reasons. The effect of the clamp is that there will be a less stressed region just under the clamp (yellow zone - 1.209 MPa), after that stress values will be approximately equal for the whole panel (1.343 MPa). Obviously, observations made for the displacements at the clamp will be valid also for the stresses.

2.4 Methodologies to decrease displacements - Question 2D

In order to decrease the displacements along the panel, two strategies are considered:

- Splitting up the panel into three parts and add more thickness to the middle portion
- Adding two stiffeners to the system

The goal is to model both options on ABAQUS and evaluate which is the most effective strategy.

2.4.1 Option 1 : Doubling the thickness of the middle part

Essentially, this optimization method involves the division of the panel into three parts and increase of the thickness of the middle part to 4 mm instead of 2 mm. The modeling process in ABAQUS is similar to the original one but in this case partitions between the parts have to be applied. However, an important step is to **assume that there is perfect bonding between the interfaces as well as no stress concentrations at interfaces**. Indeed, due to the increase of thickness of the panel, there will be geometrical discontinuities that will lead to stress concentrations at the interfaces between the thicker and thinner parts. **This stress concentrations will not be taken into account in the ABAQUS model, hence this could lead to an underestimation of the stresses at critical points**

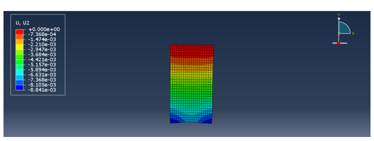

Figure 16: Displacements along the y-axis of the panel with thicker middle center

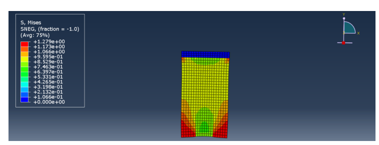

Figure 17: Mises Stresses across the panel with thicker middle center

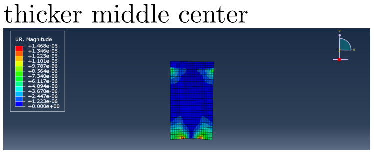

Figure 18: Rotations' magnitude through the thicker middle center panel

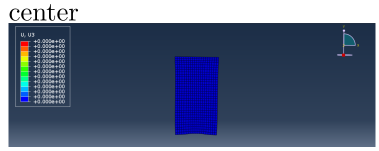

Figure 19: Displacements along the z-axis through the thicker middle center panel

- **Displacements** - According to the ABAQUS model, applying this option will lead to generally lower displacements throughout the whole panel but the displacement distributions changes slightly. In fact, the most critical parts in this case will be the bottom corners, while in the original case the part where the displacements are most critical is the whole bottom portion. Furthermore, it can be also seen that in this case displacements along z-axis, due to the effect of bending, will not be present. Lastly, rotations are limited effectively throughout the whole panel: the only rotations that will be present are at the bottom interface locations and at the top corners.

- **Stresses** - Also the stresses will be generally lower across the whole panel. In particular, it can be seen that the stress concentrations at the top corners are decreased (from 1.612 MPa to 1.173 MPa) and this time the most stressed parts are the bottom corners, while in the central bottom region the stress is lowered dramatically. This is mainly due to the increase of stiffness of the middle center: this portion will be more resistant to deformations and consequently the stress will be lower.

- **Advantages**:

  - Easy to replicate the model in ABAQUS

  - Reduces the displacements and stresses significantly

  - Limited rotations of the panel

  - No Bending

- **Disadvantages**:

  - There would be more weight in the middle part due to the increase of thickness

  - As previously said, having a thicker portion in the panel will induce stress concentrations at the interfaces

2.4.2 Option 2 : Adding stiffeners to the panel

Rather than increasing the thickness of the middle part, the second option will be concerned with the usage of stiffeners attached at 125 mm from the outer edges. Now, the main difference with the modelling process of the original case is that 3 parts need to be created (one for the panel and two for the stiffeners) and assembled effectively through the constraint "Tie". However, the main assumptions are **perfect assembly between the stiffeners and the panel, so no stress concentrations at the bonding, and stiffeners are assumed to behave as beam elements B31**. The results obtained are illustrated below.

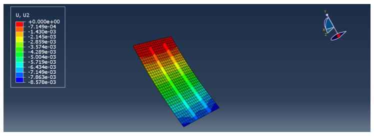

Figure 20: Displacements along the y-axis of the panel with stiffeners

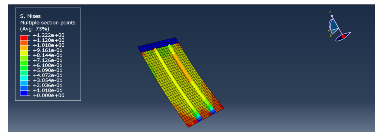

Figure 21: Mises Stress across the panel with stiffeners

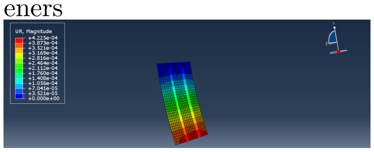

Figure 22: Rotations' magnitude throughout the panel with stiffeners

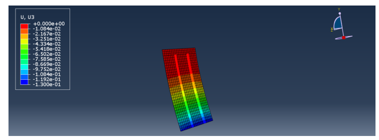

Figure 23: Displacements along the z-axis for the panel with stiffeners

- **Displacements** - Displacements along the y-axis are reduced slightly more than the panel with thicker middle center. However, to counterbalance this beneficial effect, there will be critical displacements along z-axis, due to the bending given by the stiffeners, and consequently the overall magnitude of the displacements will be greater even than the original case. Also, rotations' magnitude is increased significantly than the previous strategy.

- **Stresses** - Regarding the stresses, their values are decreased even more than the thicker middle center panel. Although stress distributions will be approximately similar between the two strategies, in this case there will be a critical location at the bottom center of the panel between the stiffeners, hence stresses are increased in that portion with respect to the previous method, but critical locations at the bottom corners are more limited.

- **Advantages**:

  - Reduces the displacements and stresses significantly

- **Disadvantages**:

  - There would be more weight due to the presence of the stiffeners

  - Relevant rotations throughout the whole panel

  - Presence of stress concentrations due to linking the stiffeners to the panel

  - Bending

2.4.3 Choosing the best option

From the results, it is now easy to choose between the two optimization strategies. If the goal is minimizing the displacements along the y-axis as much as possible, **then the most effective solution will be the one with the stiffeners**, but in this case it will be necessary to pay attention to structural integrity due to the presence of significant bending and rotations. On the other hand, if the goal is to reduce displacements along the y-axis and to maintain a good structural integrity, **then the solution with the thicker middle center will be most suitable for these kind of requirements**. Personally, if I have to select an option I will choose the first method because the displacements are reduced and equilibrium is maintained effectively.

2.5 Changing in ABAQUS model

If the panel was first clamped at the bottom at then loaded with the same weight of the first case, **the boundary conditions of the bottom clamp will not be no longer encastre**. This because, otherwise, zero displacements along every direction of the local coordinate system will be achieved. In order to obtain non-null results, the boundary conditions of the bottom clamp have to be modified.