ANALYSIS OF SOLAR PANEL SUPPORT STRUCTURES

[1]A. Mihailidis, [1]K. Panagiotidis, [1]K. Agouridas[*]
[1]Lab. of Machine Elements & Machine Design, Dep. of Mechanical engineering, Aristotle University of Thessaloniki, Greece

KEYWORDS
Solar array, frame structures

ABSTRACT
The use of renewable energy resources is increasing rapidly. Following this trend, the implementation of large area solar arrays is considered to be a necessity. Several design approaches of the supporting structures have been presented in order to achieve the maximum overall efficiency. They are loaded mainly by aerodynamic forces. International regulations as well as the competition between industries define that they must withstand the enormous loads that result from air velocities over 120 km/h. Furthermore, they must have a life expectancy of more than 20 years. In this paper, the analysis of two different design approaches of solar panel support structures is presented. The analysis can be split in the following steps.
1. Load calculation, which includes the creation of a simple CFD model using ANSA as pre-processor and ANSYS-CFX as solver to determine the pressure distribution on the solar panel area and the application of EUROCODE 1 to determine the resultant magnitude of the forces acting on the surface of the solar panels.
2. Analysis of the structure, which includes the creation of a FE model using ANSA as pre-processor. Loads calculated in the first step are applied to the model. As solver MSC Nastran is used.
3. Identification of the structure critical points. According to the results weak points are redesigned in order to increase the endurance.

1. INTRODUCTION, SUPPORT STRUCTURE DESIGNS

Nowadays the demand for clean, renewable energy sources is increasing. In order to collect solar power effectively, it is necessary to use large areas of solar panels properly aligned to the sun. A wide variety of design solutions is suggested so as to achieve maximum efficiency. In this paper the analysis of two different design approaches are presented:
1. A fixed system that is mounted to a certain position as shown in Figure 1. The orientation of the solar panel array is adapted to the installation site so that the efficiency of the system is optimized.
2. An adjustable system that features mechanisms to enable it to be automatically rotated around 2 axes as shown in Figure 2. This system has the advantage that light beams are all day long normal to the surface of the panels.

The fact that these structures have to support a large area of solar panels (in both structures the area is about 50m^2), makes them vulnerable to wind action. Laws and regulations prescribe that such structures must withstand air velocities over 120 km/h. Competition among industries raises this limit to 140 km/h.

2. LOADS – BOUNDARY CONDITIONS

The main load of the support structures is caused by the wind action. Wind load has to be calculated according to EUROCODE 1 (1). According to this regulation only the total wind force is determined, and therefore it cannot be applied to a FE model directly. It has to be distributed to node loads. On the other side, a CFD model results in the pressure distribution that enables one to determine easily the required node loads. However, the total aerodynamic force is different from that obtained by EUROCODE. The reason for this deviation is that EUROCODE considers several effects such as season of the year,

installation site, required service life, and so on, whereas most CFD models do not. Consequently both EUROCODE and CFD model must be used.
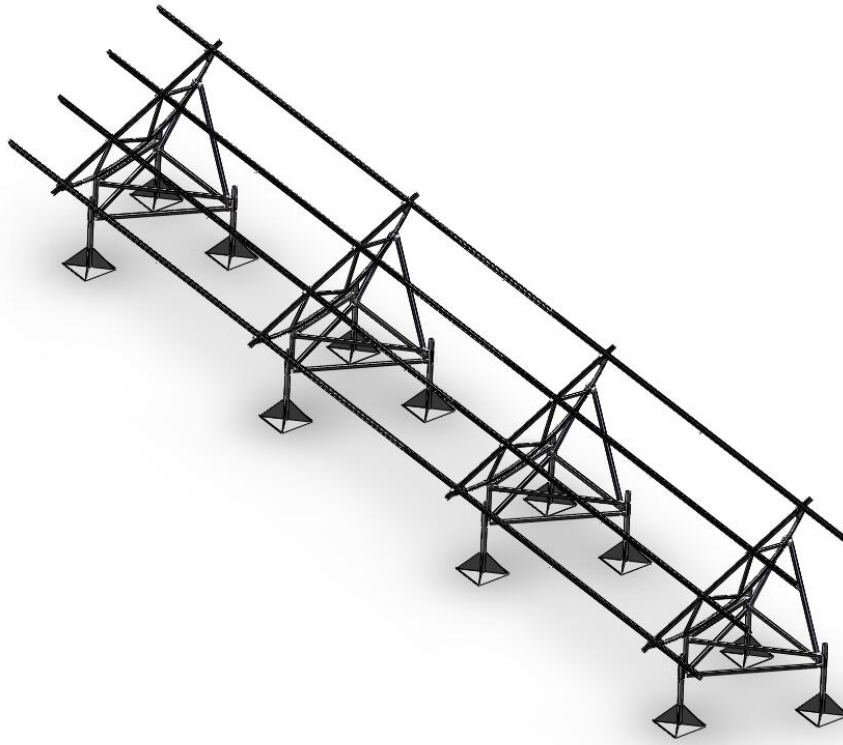
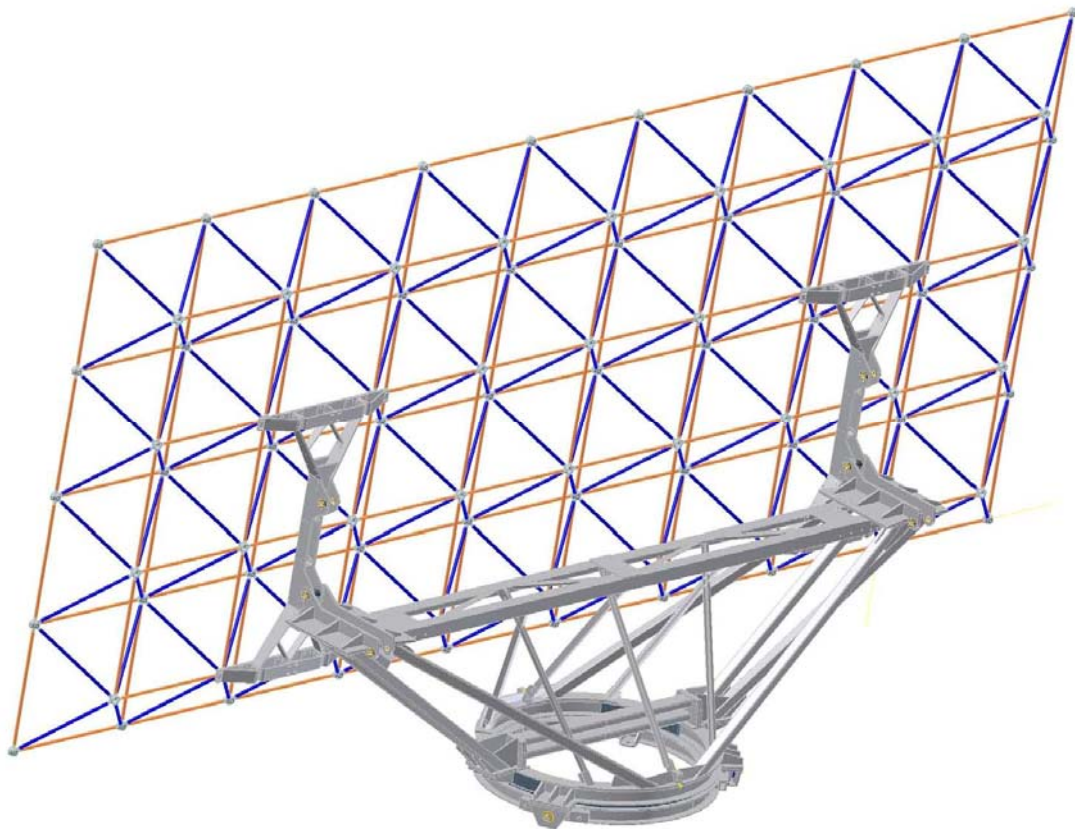Figure 1 – Design A: Fixed support structure design (EXEL MAKMETAL)

Figure 2 – Design B: Adjustable support structure design (IRIS - PTOLEMEO)

Load calculation

Wind direction is stochastic and therefore it is necessary to compute the pressure distribution for a variety of wind directions, because it is usually very difficult to estimate which one is the most critical. In the current analysis the following wind directions are considered 0, 15, 165 and 180 degrees. Figure 3 shows the CFD model and the boundary conditions for wind direction angle 0 degrees.
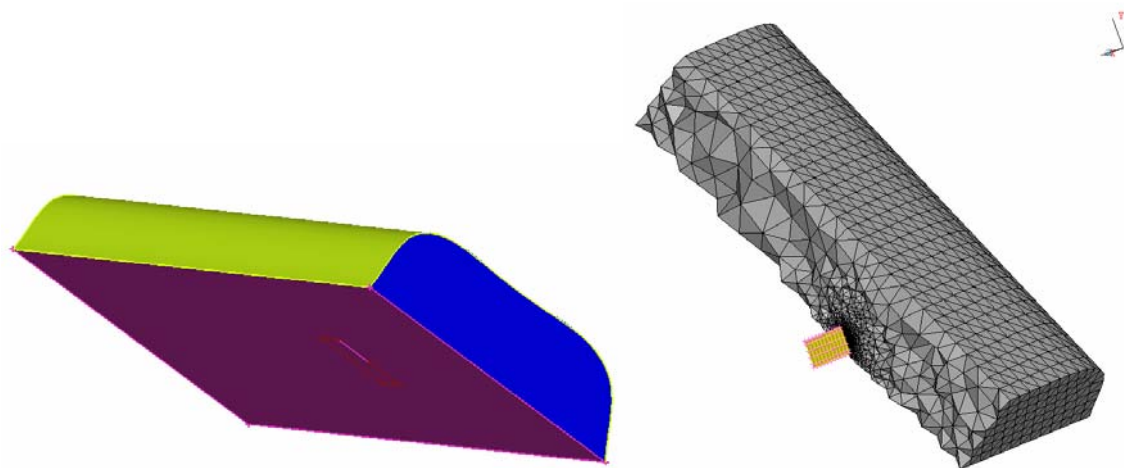
Figure 3 – Fluid model, boundary conditions:
      1 Purple: Ground and plate
      2 Green: Opening where the relative pressure is 0
      3 Blue: Given velocity with vector normal to the control volume surface

The aerodynamic loads are caused mainly by the solar panel array whose thickness is very small regarding its other dimensions. Therefore, it can be modelled as a thin plate consisting of shell elements in a control volume. The dimensions of the control volume are chosen large compared to the dimensions of the plate. The model was solved using the ANSYS-CFX. Typical results are shown in Figures 4 and 5.
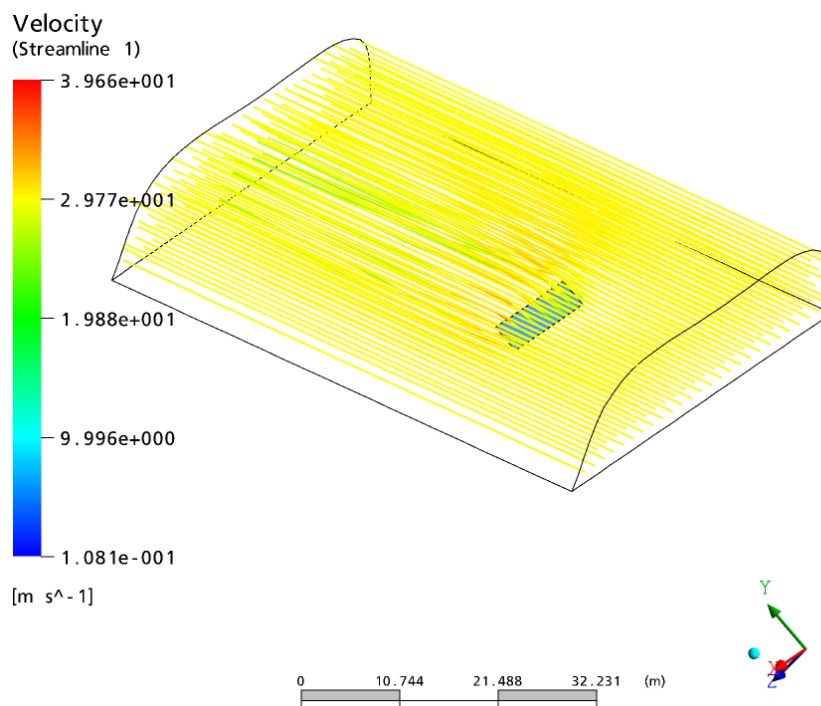
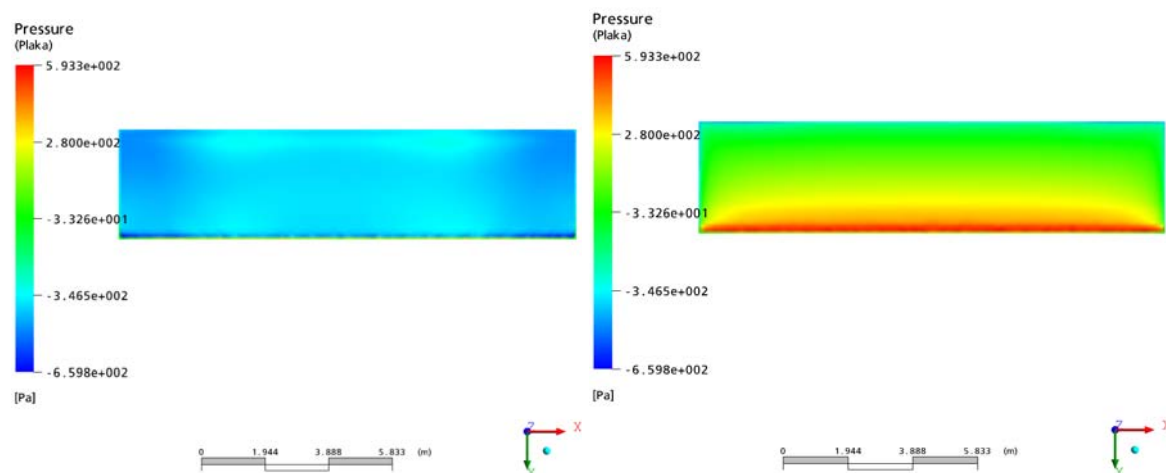Figure 4 – CFX Results: Air Streamlines

Figure 5 – CFX Results: Pressure distribution on the front and the back of the plate

From these results the pressure distribution is obtained. Following, the resultant force and the node loads can be determined.
The resultant force results from the summation of the loads acting on each node.

$$F_{CFX} = \sum_{i=1}^{n} \left(F_{CFXbi} + F_{CFXfi} \right)$$

where:

F_{CFX} Resultant force obtained from CFD model

F_{CFXbi} Force on node i of the back side of the plate

F_{CFXfi} Force on node i of the front side of the plate

n Number of nodes

As mentioned earlier the total force according to EUROCODE F_{Eu} differs from that obtained by the CFD model. A scaling factor C can then be calculated.

$$C = \frac{F_{Eu}}{F_{CFX}}$$

The actually applied forces on the nodes are:

$$F_{node} = C \left(F_{CFXbi} + F_{CFXfi} \right)$$

The procedure to import forces to the static model is the following:

1. Export forces from ANSYS-CFX
2. Export the geometry of the plate including the mesh from ANSA (2). The mesh must be retained so that the position and the number of the nodes remain the same
3. Process of the forces by adding the force at the back and at the front node of the plate scaled as mentioned above.
4. Compose the NASTRAN load file
5. Import the NASTRAN load file into the static FE model.

3. STRUCTURE MODELLING

Since both structures designs consist only of thin-walled parts the 3-d solid parts imported from the CAD file must be first translated to surfaces as shown in Figure 6.

Triangular and rectangular shell elements (CTRIAS and CTETRAS) (3) are used to model the resulting surface.
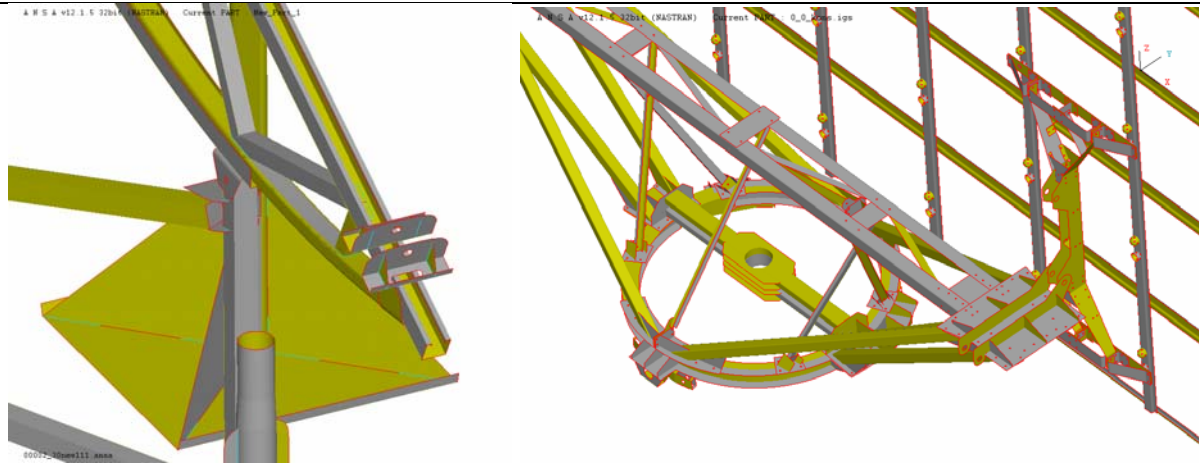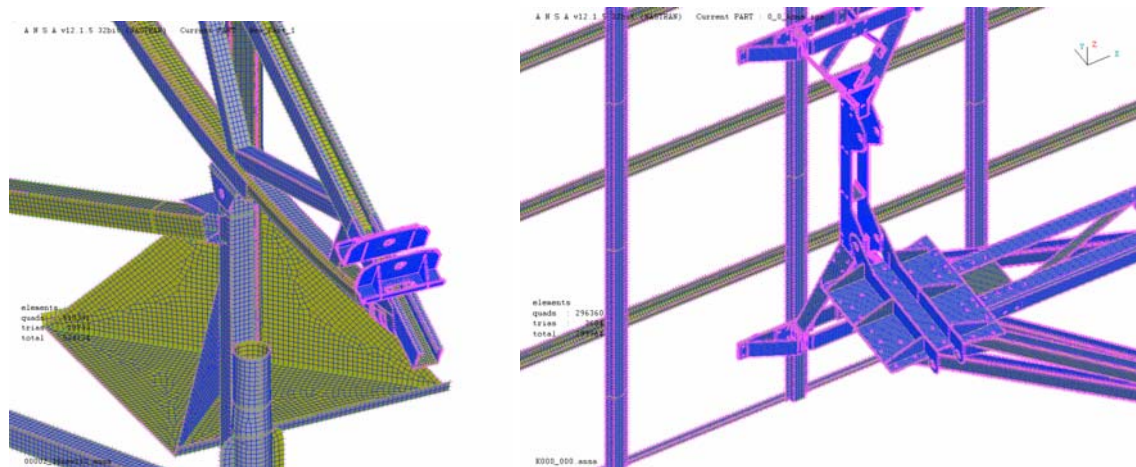
Figure 6 –Solid parts translated to surfaces

Figure 7 – Shell mesh

Figure 8 shows the modelling bolted connections. RB2 elements are used to constrain the holes with the centre node and a CBAR element to model the body of the bolt. Since bolted connections cannot support rotational loads, the torsional moment of inertia of the CBAR is set to zero.
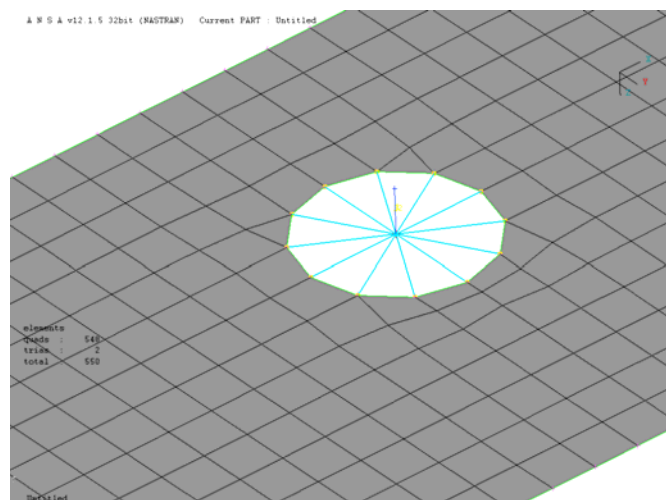

Figure 8 – Typical bolted connection

The space frame of Design B consists of nodes and tubes. The nodes are modeled as shell elements and tubes as CBAR elements shown in Figure 9.
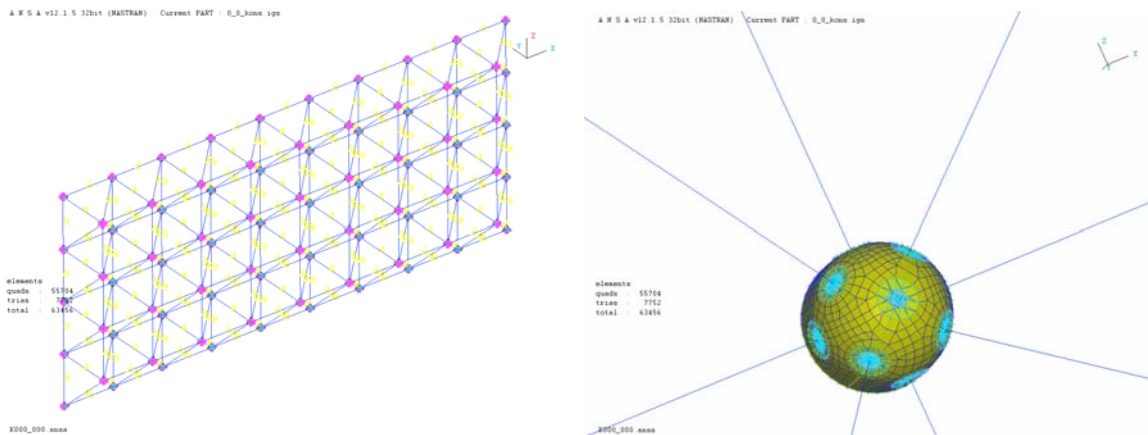
Figure 9 – Space frame

Critical welds are modeled according to the Hot Spot Stress analysis (4) as shown in Figure 10. This analysis requires that the stresses at certain positions are known, in order to extrapolate the results and obtain the Hot Spot stress at the welding.
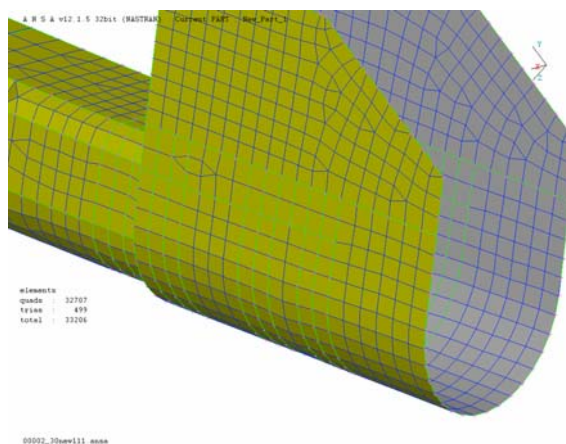

Figure 10 – Mesh used for Hot Spot Stress analysis

In modelling design A self-weight can be neglected, since it is supported by many rods inserted in the ground. However the elastic properties of the ground have to be taken into account. Depending on the installation site they must be determined by geotechnical analysis.
Typical elastic properties are given in Figure 11.
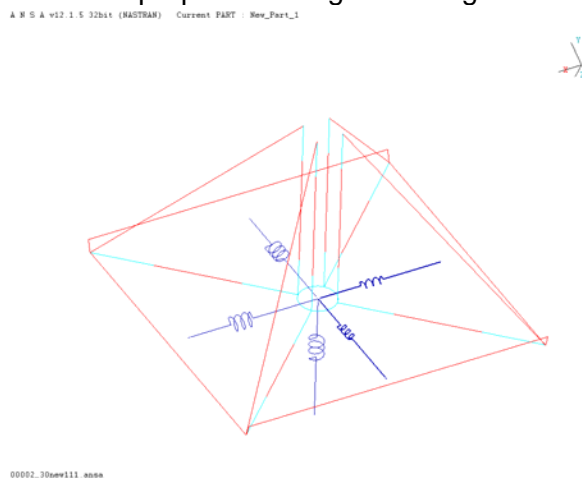
Vertical stiffness	7000 kN/m
Horizontal stiffness	875 kN/m
Torsional stiffness	120 kNm/rad

Figure 11 - Constrains of the base of design A

In design B the solar panels are attached to a space frame. It is connected by two hinges to an auxiliary frame. In this way it can be rotated around the horizontal axis. The complete support structure rest on three rollers in a circular guide. In this way it can be rotated around the vertical axis. Calculations were carried out for several angles for both horizontal and vertical axes. Moreover, the weight cannot be neglected in this design.
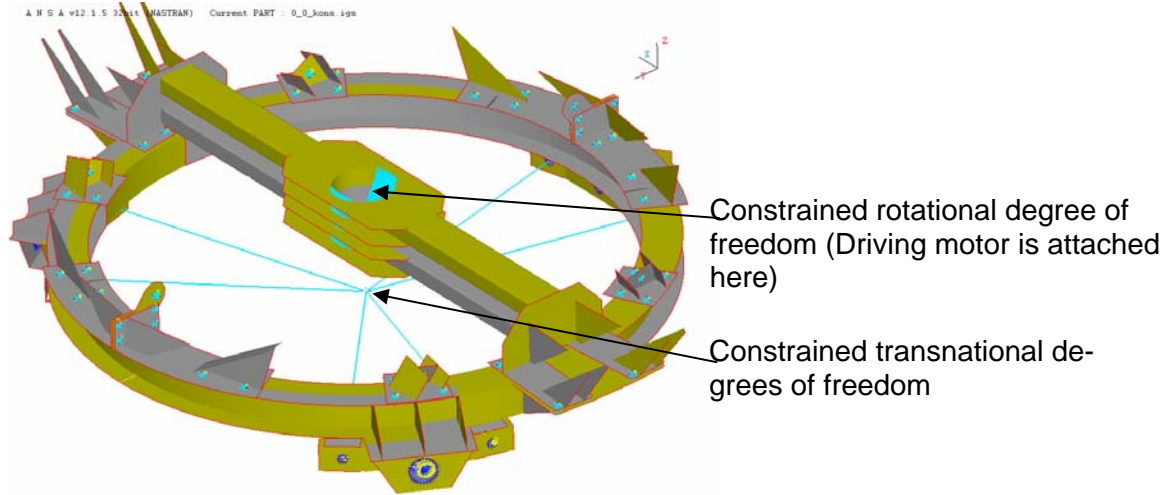
Constrained rotational degree of freedom (Driving motor is attached here)

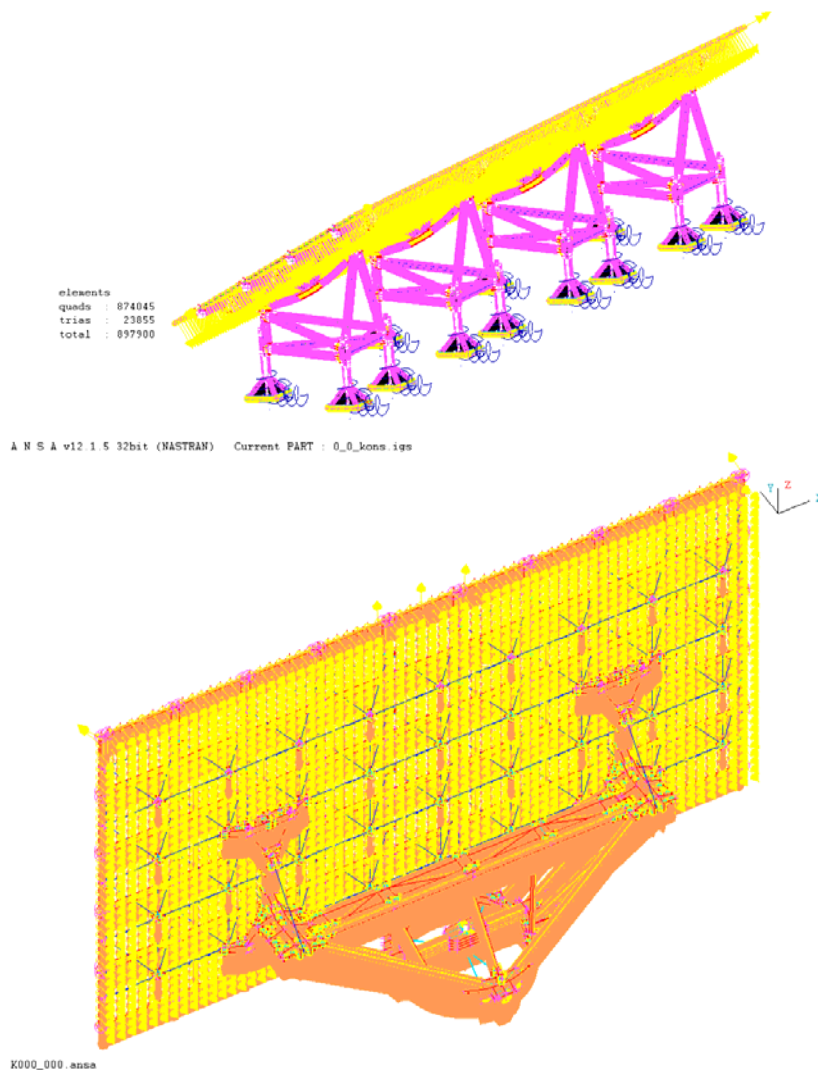Constrained transnational degrees of freedom

Figure 12 – Constrains design B

Figure 13 – Final models ready to be solved

4. RESULTS

Both models were solved using MSC Nastran. The solution types chosen initially were 101 – Linear Static and 106 – Non Linear Static taking into account large deformations and assuming linear elastic material behaviour. The comparison showed that the non-Linear solution resulted in only insignificantly better accuracy but required considerably more computing time. Therefore, the Linear Static solution was chosen.
The assumption of a linear elastic material behaviour is justified by the fact that the allowable maximum Von-Mises stress is set far below the yield stress, because the structure must withstand the highly dynamic loads that actually occur.
The following Figures 14 – 18 show some problems of the initial designs and how they were resolved.

Figure 14 shows the initial design of the support of a longitudinal frame member. Since it is fixed, the resulting stress field includes impermissible high values. In the improved design shown on the right of the Figure 14, the maximum stress is significantly reduced, by fixing the longitudinal member to a U – formed attachment by a single pin.
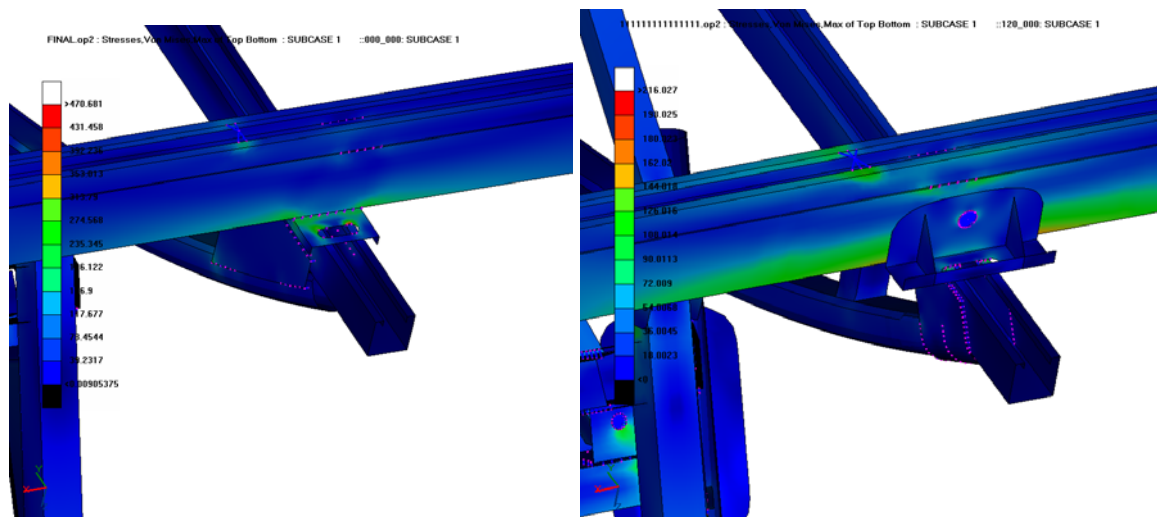

Figure 14 – Frame member support

The maximum stress in the transverse frame member was significantly reduced by a finer triangulation near the end points as shown in Figure 15.
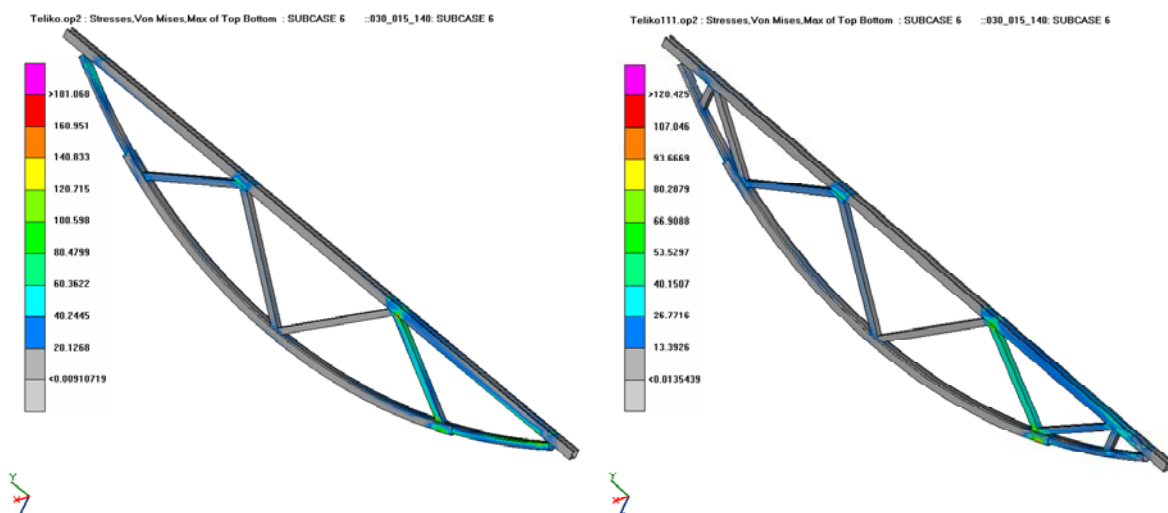

Figure 15 – Triangulation of the transverse frame member

Figure 16 shows the reinforcements that were necessary to increase the load caring capacity of the auxiliary frame design.
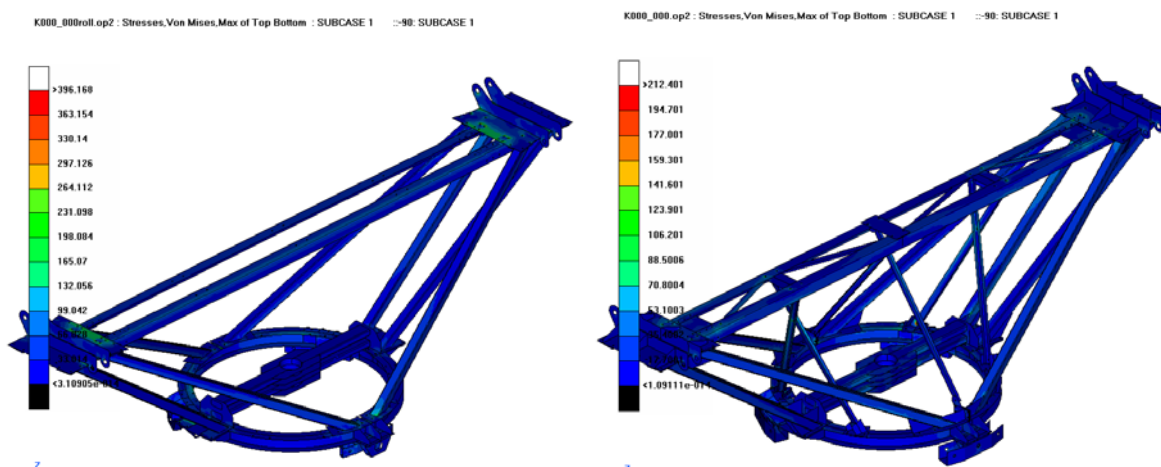

Figure 16 – Reinforcements of the auxiliary frame

Figure 17 shows that the maximum stress be could significantly reduced by adding properly designed reinforcements.
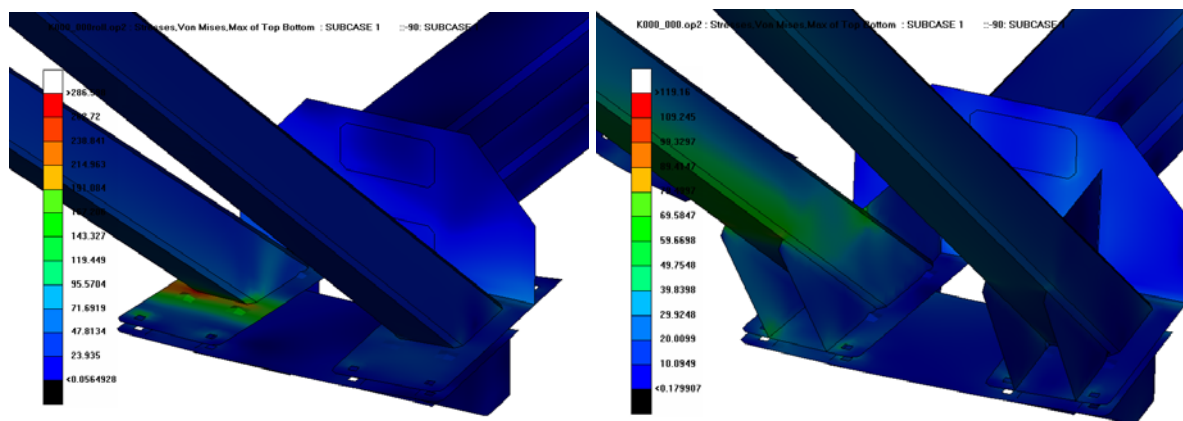

Figure 17 – Reinforcements of weldings

The space frame of the design B consists, as mentioned above, from nodes and tubes. Tubes can only be loaded by tensile or compressive forces. In the later case Von-Mises stress as yield criterion is not enough for this part of the structure and buckling calculation is also necessary.

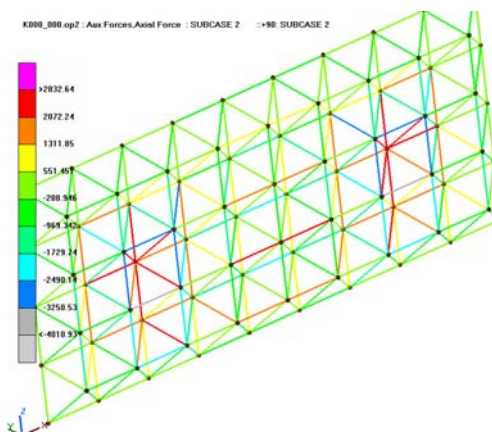Figure 18 shows the forces used for buckling calculation of the tubes.

Figure 18 – 1d Element forces used for buckling calculation
The following figures 19 and 20 show the deformation and the stress field of the final designs of both structures.
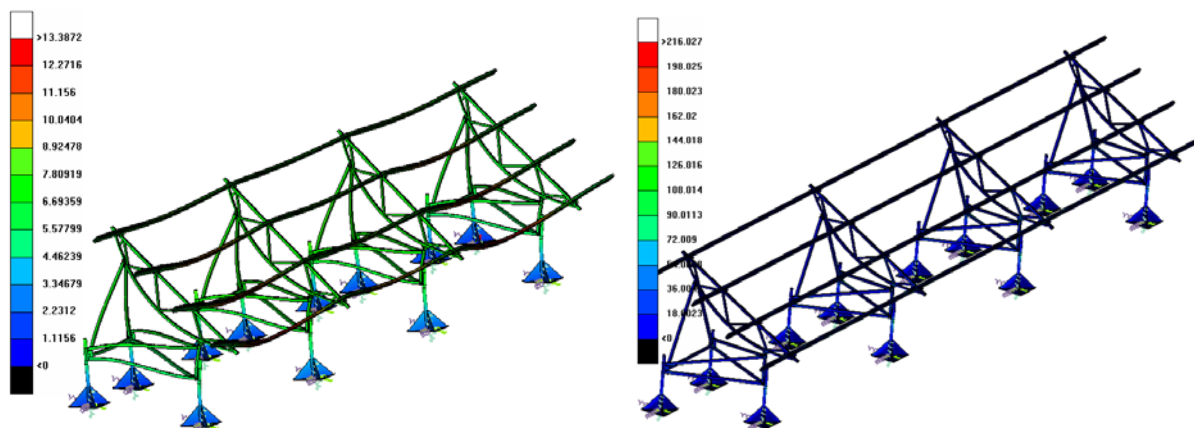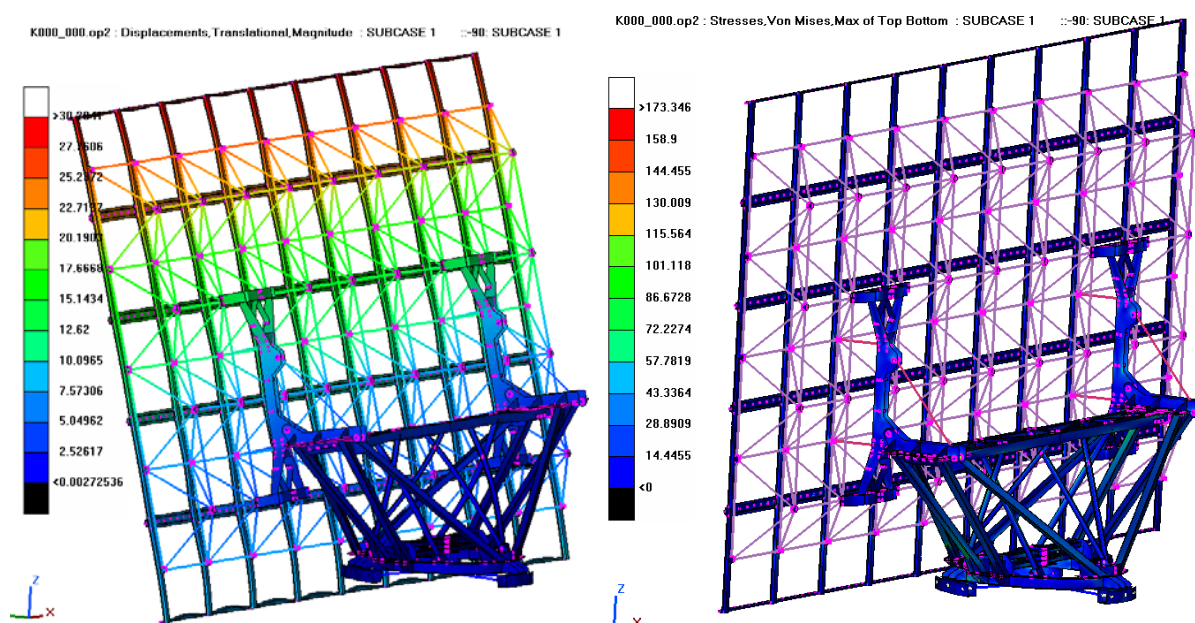

Figure 19 – Deformation and stress field of design A

Figure 20 – Deformation and stress field of design B

5. CONCLUSIONS

Even fixed solar array support structures have sofisticated design, that needs to be analyzed and often improved in order to withstand the wind load. The same applies of course to adjustable designs to an even greater extend. The analysis has to be carried out for many wind directions. Therefore, it is very important to employ productive software tools. The designs presented in this paper were analyzed using ANSYS – CFX, ANSA, MSC NASTRAN and META (5) in a very efficient way. Further, it was possible to omit the time-consuming non-linear solutions, because comparative calculations showed that the accurancy gain obtained by them is insignificand.

Figures 21 and 22 show the installed solar arrays.

Figure 21 – Design A

Figure 22 – Design B

REFERENCES

(1)   Eurocode 1: Actions on structures - General actions - Part 1-4:Wind actions

(2)   ANSA version 12.1.5 User's Guide, BETA CAE Systems S.A., July 2008

(3)   MSC.Nastran 2005 r3,Quick Reference Guide

(4)   International Institute of Welding, Recommendations for fatigue design of welded joints and components, June 2005

(5)   μETA Postprocessor, version 6.2.0. User's Guide, BETA CAE Systems S.A., June 2008