Paper published in: Structural Engineering International, No. 4, pp292-297, 1999.

LARGE PREFABRICATED THIN FACED SANDWICH ELEMENTS FOR ANTENNA COVERAGE

Prof. Dr. M. Farshad
P. Flüeler, Dipl. Ing.
Swiss Federal Laboratories for Materials Testing and Research,
EMPA, Dübendorf, Switzerland

Summary

Included is the result of an engineering work related to realization of prefabricated sandwich elements for antenna coverage in the telecommunications system on mount Säntis in Switzerland. A spectrum of activities including setting up of technical requirements, conception of structural system, choice of appropriate material and structural elements, testing of material components and structural elements. The side dimensions of a prefabricated element was to 6 m and 6.6 m, the basis diameter of the middle shell was 5 m, its rise was of 600 mm and the thickness of sandwich element was about 101 mm and thickness of the two facings less than 0.5 mm. The experimental investigation included material, hail, bending, and wrinkling tests as well as testing of a whole structural element. In the later tests, tsandwich element was subjected to statically simulated wind pressure and wind suction. The theoretical work included analytical modeling and Finite Element simulations. An account of supervision of of production as well as transport and installation of the prefabricated elements is also given.

1. INTRODUCTION

The telecommunications system on the mount Säntis in Switzerland (approximately 2500 meter over the see level) consists of a complex system of antenna, tower, and general-purpose installations. A project dealing with the enhancement of this system entitled Säntis 2000 was initiated by Swisscom. These changes influenced the structural system consisting of antenna coverage and its constituting elements. The existing structural elements covering the antenna had different shapes; one of the typical structural elements had side dimensions of 6 m x 6.6 m. Elements of this type were flat plated all around, but were supplemented by a shell in their middle. The shell was a sector of a dome and had a basis diameter of about 5 meter and a rise of about 60 centimeter. According to the architectural planning, the existing geometrical shape of the elements should also be adopted for the new elements.

In the development and realization of this project were a variety of sources and specialties involved. These included the architects, the engineering firms, the material producers, and the specialists in preforming the foam sectors and putting the sectors together and laminating the sandwich structure. The prefabricated elements were transported and installed on a steel structural framework. The role of the Swiss Federal Laboratories for Materials Testing and Research (EMPA) was to act as a neutral consultant in the conception, the testing, the engineering analysis, and in supervision during the realization of this part of the project.

2. REQUIREMENTS ON THE MATERIAL AND THE STRUCTURAL SYSTEM

Following functional requirements on the antenna coverage for the telecommunications system were set:

(1) Transparency for radioelectrical signals up to 20 GHz
(2) Appropriate short as well as long term mechanical and thermal material properties
(3) Adequate mechanical resistance and stability against static and dynamic loading including wind, hail, ice and snow, and thermal gradients
(4) Appropriate connections, which provide structural strength and watertightness

(5) Reparability
(6) Appropriate environmental properties (fire, biological influences)
(7) Feasibility of production, transport and installation in one piece

These requirements would provide a set of guidelines for the choice of the material and the structural system. A number of these criteria were assessed by comparison of the material specifications, material and element tests and by structural modeling and simulation.

3. SANDWICH CONSTRUCTION AND ITS COMPONENTS

To arrive at an optimal choice for the appropriate antenna cladding, exploratory studies and material tests were performed. These investigations were supplemented by the one-site examination of the previous cladding, performance of the candidate materials in similar situations; criteria on the material and the structural system, and the production feasibility. Following these studies, a choice based on using a sandwich element with foam as it core was made. AS other types of sandwich constructions, the facing would provide additional strength and stiffness; it would also act as a protective layer against the environmental influences such as water, ice, and hail. However, the thickness of the facing was to be very small, so that the radioelectrical transparency would not be reduced.
The sandwich construction chosen for this purpose consisted of foam as the core, two facings, and a protection layer, which would be applied at the outside surface. The core was integral Polyetherimide (PEI) foam, the density of which was higher at both sides in comparison with the density in the middle of the foam. It varied from both sides to the middle of the foam thickness and ranged from 53 kg/m3 (in the middle) to about 67 kg/m3 at the outer zone. The thickness of the core amounted to 100 mm. The face sheets were made of two plies 0°/90° (US Glass 299 g/m^2) with a nominal thickness of 0.35 mm. The two facings were laminated directly on the foam core resulting in a measured thickness amounting to about 0.4 mm. On the outer side of the element, a Polyurethane (PUR) lack was applied; this was intended as surface finish and UV protection at the weather side.
The 6-m x 6.6-m vertical sandwich elements were additionally reinforced along the edges by a Laminated GFRP profile and were then placed in a ring of steel profile, which would act as a transport frame and later on connection frame. The structural elements were to be continuously connected to the steel framework by means of the above mentioned steel U profile. In addition to the vertical elements, other structural elements with the same sandwich construction were planned; these elements would be placed at the vertical and horizontal corners of the antenna coverage system.

4. EXPERIMENTAL INVESTIGATION

To investigate the suitability of the material and the sandwich structure for the requirements, which were outlined before, a number of tests, were performed on the constituting components of the sandwich section and on the sandwich section as the whole. Additionally, a full-scale test was carried out on a prototype of the whole sandwich element with real structural boundary connections. This section summarizes the material and the structural tests. These experimental investigations were instrumental in the decisions leading to the choice of the material as well as the whole system.

4.1. Material investigation

To compare various candidate materials, a number of mechanical and physical tests were performed. Physical tests included the porosity and density measurements on the foam and measurement of thermal extension of the foam and the GFRP facing. The mechanical tests included tensile tests on the GFRP facing, shear tests on the core, peal-off tests on the sandwich material. They also included hail and fire tests on the sandwich section.

4.2. Bending tests on sandwich beams

Samples of sandwich beams made of various combinations of foams and facings were subjected to four point bending tests. The beam samples were 2.4-m long, 300 mm wide, and had the height equal to the sandwich section, i.e., a thickness of about 100 mm. Fig. 1 shows the experimental arrangement for the four-point bending tests and the geometric parameters involved in the tests.

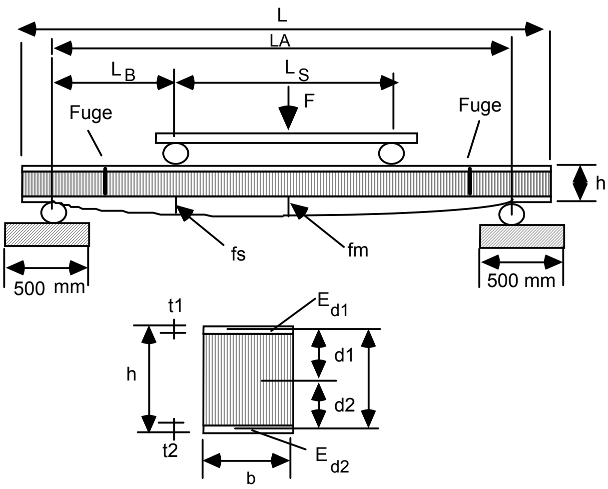


Fig.1: The experimental arrangement and the geometrical parameters in the four points bending tests on sandwich beams

The results of four point bending tests are summarized in table 1. This table shows the maximal failure bending load and the corresponding maximal lateral deformation. As we note, the failure of the thin-faced sandwich beams was initiated by wrinkling of the facing at the compression side of the beam, i.e., at the topside of the element.

Table 1: Results of bending tests on the sandwich beams

Sample No.	Maximum force [N]	Maximum deflection(3) [mm]	Failure mode
1 (1)	4'740	49.0	compression failure at the top facing due to face wrinkling occurred in the zone between two loading
2 (1)	4'450	47.0	2 compression failures at the top facing due to face wrinkling near the supporting zone and the zone between two loading
3 (2)	5'000	53.0	compression failures at the top facing due to face wrinkling near the supporting zone

(1) Top face (compression side of the beam): outside face of the structural element (2 layers of GFRP facing and PUR lack)

(2) Top face (compression side of the beam): outside face of the structural element (2 layers of GFRP facing)

(3) Total value of bending deflection in the middle of the beam

4.3. Wrinkling of thin faced sandwich panels

In classical application of sandwich constructions, typical ratio of the thickness of facing to that of the core is 5 % and even more. In contrast to this conventional sandwich construction, in the present application, sandwich elements with very thin-faces having facing to core thickness ratio of 1 % and less were to be used. Here, the number of facing layers was two; this is to be compared with conventional sandwich constructions in which the facings are composed of more than 10 plies. Due to the small number of facing plies, occurrence of facing imperfection during production and susceptibility of such elements to local damages is expected to be more than in the conventional laminated materials. In particular, wrinkling of faces under compressive forces is a phenomenon, which can not only occur because of the thin facing, but can be drastically influenced by imperfections. Wrinkling of faces can affect the lifetime of sandwich elements. Structurally, it signifies reduction in the stiffness; functionally it can cause penetration of water into the core.

To investigate the wrinkling resistance of the thin faced sandwich elements; an experimental and theoretical wrinkling study was carried.

4.3.1. Samples for wrinkling experiments

Direct compression tests were performed on different sandwich sections of various candidate thin-faced sandwich panels. Various sandwich elements tested in this work had Polyurethane (PUR), polyetherimide (PEI) foams as core and aluminum, aramide, or two layers of 0°/90° of GFRP as their facings. The overall dimensions of the sandwich samples investigated are shown in Fig. 2, while the actual measured Young's Modulus of the facings is given for different fiber orientations in table 2. The Young's modulus was determined by the tensile tests. Prioer to wrinkling tests, tensile tests were performed on the facing laminate. These tests resulted a Young's modulus of 20700 N/mm^2 in the direction prallel to the vertical edge and a value of of 18700 N/mm^2 in the direction prallel to the horizontal edge of the compression panel.

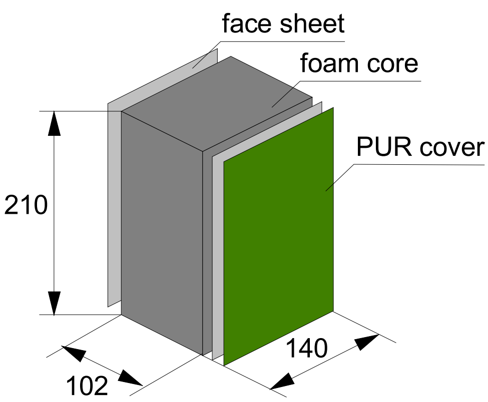
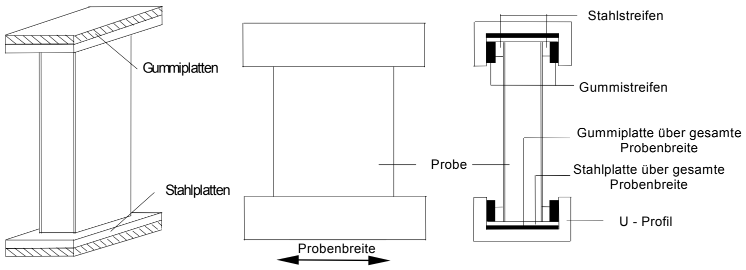


Fig. 2 Overall dimensions of sandwich samples for wrinkling tests

4.3.2. Wrinkling experiments

The experimental set-up for the direct compression test is shown in Fig. 3. The sandwich sample was located between two pressure plates and was centered at each end with two adjustable rubber plated guides. The load cell of the testing machine was fixed to the lower pressure plate and the cross-head acted via a ball joint on the upper pressure plate. Thereby, effects due to a possible misalignment of the two pressure plates and premature failure at the two end regions were avoided. To monitor the wrinkling event, strain gauges were installed at both sides of the samples in the longitudinal and transverse directions. Additionally, to monitor the post-wrinkling event, the method of Moiré photography was used.



U-Profil :	Aluminium (120x65x10 (8)) L = 196 mm
Streifen :	Stahl lxbxt =185x20x4mm Gummi lxbxt = 200x28x5.50mm
Platten:	Stahl lxbxt = 301x98,5x8mm Gummi lxbxt = 305x100x6mm

Fig. 3: Experimental arrangement for wrinkling tests

4.3.3. Results of wrinkling tests

Fig. 4 shows the wrinkling patterns of a sandwich panel (sample No. 6). This panel had an integral PEI foams as the core and two layers of GFRP as its facings. The dimensions of this panel were H = 280 mm, B = 182 mm, t_f = 0.4 mm, c = 80 mm. The maximal wrinkling load of this sample amounted to F_{max} = 12'300 N.

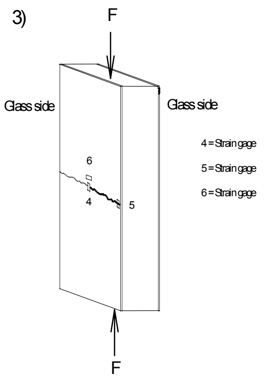


Fig. 4: Wrinkling pattern of sandwich panel No. 6 (PEI core and GFRP facings)

In order to investigate the effect of the fiber orientation of the facings on wrinkling behavior, three groups of sandwich samples were tested. In the first two groups, the fiber orientations of the GFRP facings were parallel or perpendicular to the sample edges (0°, 90°); in the third sample group, the facings fibers were orientated at an angle of ±45°. The maximum compression load and the of the sandwich sample at failure amounted to 17000 N in the 0°/90° direction and 16000 in the 45° direction. The corresponding failure strain was 0.95% and 2.19%. respectively.

The wrinkling event and the post-wrinkling pattern was monitored by the Moiré phonographic technique as well as with the strain gauge measurements. Fig. 5 shows the wrinkling deformation profile of a sandwich panel (sample No. 8) made of integral PEI foam as its core and two lacers of GFRP as its facings. This Figure shows the deformation profile in the wrinkling zone. The eight curves in this Fig. represent the lateral deformation of the facing with increasing axial load. In this Fig., the horizontal axis represents the length coordinate; hence the number 400 signifies the whole length of the panel.

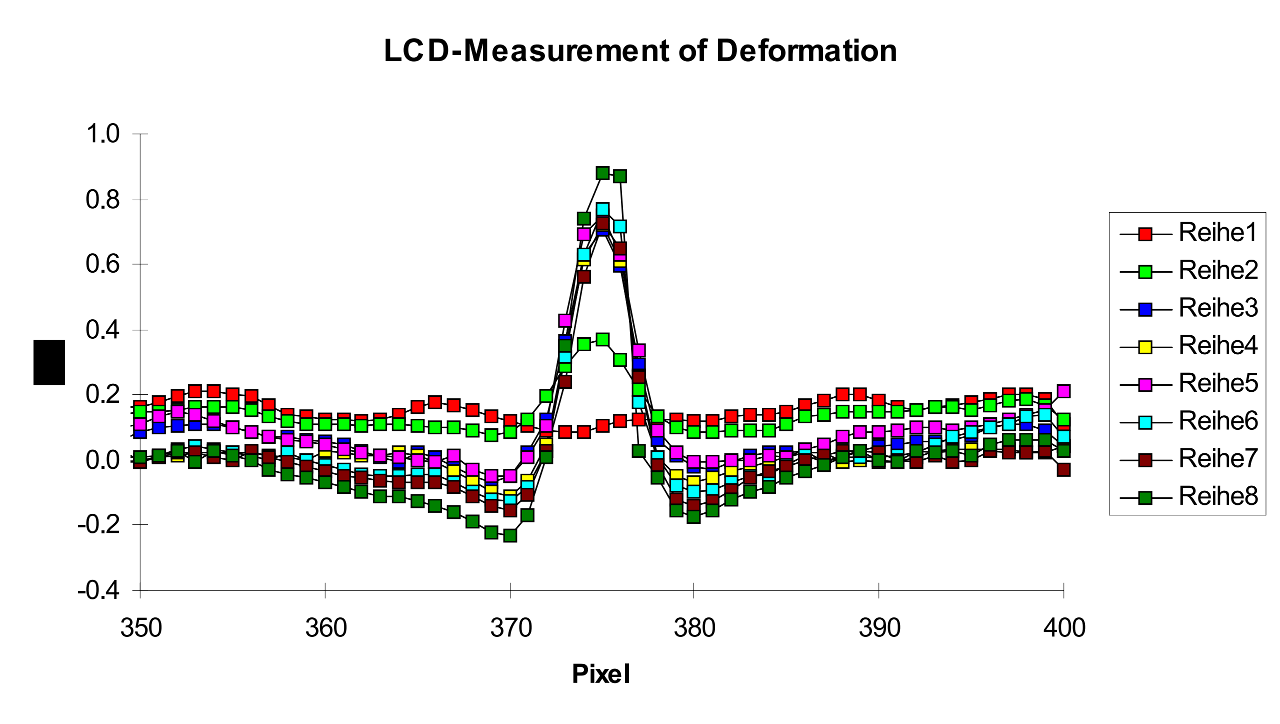


Fig. 5: Lateral deformation profile of the facing of sandwich sample No. 8 under axial compression obtained by the Moiré photographic technique. The Legend including the word "Reihe" designates the sequence of Moiré readings.

5. TESTING OF THE WHOLE SANDWICH ELEMENT

5.1. Description of experiment

Static wind pressure and wind suction experiments were performed on the sandwich structure described before. The wind pressure tests simulated a wind velocity up to 230 km/h. The maximum distributed pressure on the element corresponding to this loading would amount to 49 mbar. The wind suction tests simulated a wind velocity of about 190 km/h. The maximum distributed negative pressure on the element corresponding to this loading was 33.6 mbar. Dynamic wind loading was also planed, but due to financial limitations was not carried out. In addition to static wind pressure and wind suction tests, the natural frequency of sandwich structure was measured by the ambient vibration method.

The flat plate sandwich element tested had 6 m x 6.6 m as its side dimensions and a dome with a basis diameter 0f 5-m and height of 0.6 m at its center. The thickness of the element amounted to about 101 mm. The cross section of the sandwich construction consisted of a 100 mm integral Polyetherimide (PEI) foam and two facings; each facing was composed of a two layered 0°/90° GFRP laminate glass fiber with Epoxy matrix. Each ply consisted of a texture made of 90% glass fiber in one direction and 10% in another direction. Thickness of each ply amounted to about 0.33 mm. Prior to placing the plies on the foam; a 545-epoxy primer was applied to the surface of the foam. Adhesion of the facing was facilitated with the help of vacuum and tempering stimulated hardening of the facing laminate. At the weather side, i.e., on the convex side of the element, a brawn-yellow color Polyurethane (PUR) lack was applied. All around the edges, the element was reinforced with a U profile made of a GFRP laminate. The laminate was encased in a steel frame made of U profile. Between the steel frame and the GFRP edge reinforcement, a zero tolerance was assumed.

5.2. Test set-up for experiments on the whole element

The sandwich element was placed on a horizontal floor and was fastened at all sided on the steel frame quite similar to that of the steel structure, which was used on the construction site. The wind pressure on the element was produced by a vacuum mechanism, which acted on the inside of the element, i.e., in the space between the element and the airtight floor. The pressure underneath the plate simulated the wind suction.
The instrumentation used in testing of the prototype element consisted of strain gauges, inductive displacement gauges, dial gauges, temperature gauges, pressure gauges, and acoustic emission sensors. The strain gauges and the acoustic emission transducers were installed on the critical zones, which were a priori determined by the Finite Element simulation of experiments. In addition to local deformation measurements, a portal frame was constructed on which a movable measuring apparatus containing an inductive gauge was installed. This Part would move along the middle section of the structural element and would continuously measure the original and the deformed profile of the sandwich element. Upon the completion of tests, the method of lock-in thermography was used to assess the possible delaminations in the test plate.
In the wind suction loading, the pressure was increased from zero to the maximal value of 0.049 bar. The sequential loading and unloading took place in preplanned stages, so that the acoustic emission signals and the possible damages could be monitored. The wind pressure loading was simulated through a vacuum which was applied at the inside space of the plate, i.e., between the plate and the air tight floor and edges. Fig. 6 shows the test element on the testing floor. At this stage, the above mentioned measuring devices are not installed. The transducers, which were used for determination of the natural frequencies of the element with ambient vibration method, are shown in this figure. Fig. 7 shows the outer and the inner projective view of the element and the position of the strain gauges.

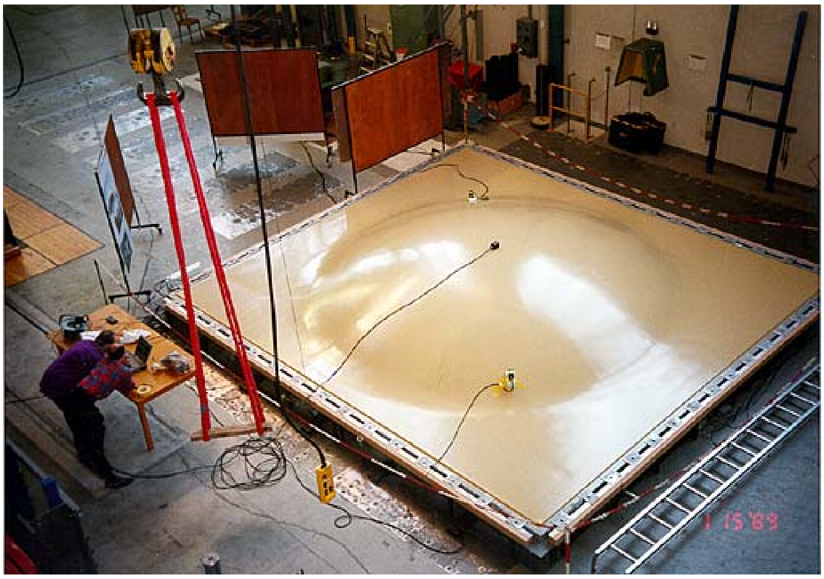


Fig. 6: A photograph of the test plate. The photo shows the plate, the edge connections, and the transducers, which were installed to measure the natural frequency of the plate with the ambient vibration method.

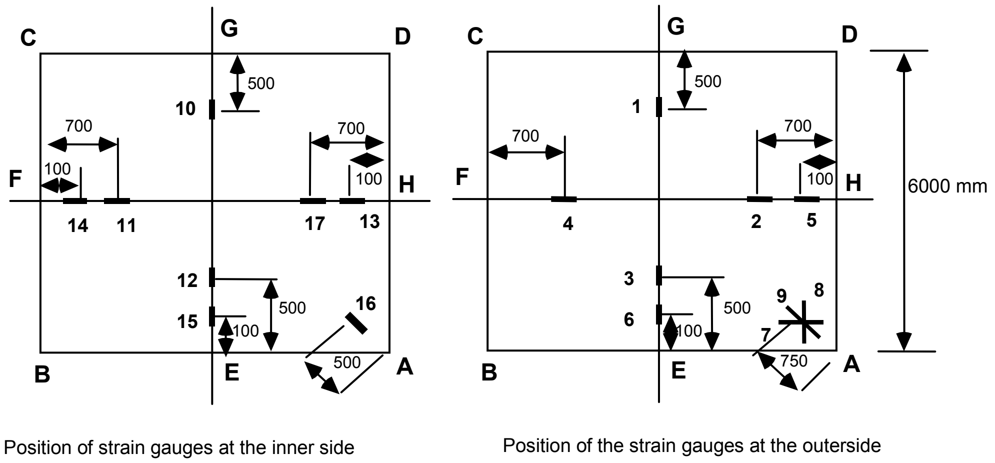


Fig. 7: Arrangement of the strain gauges at the inside and the outside of the test element

5.3. Results of tests on the whole sandwich element

5.3.1. Deformation

The sliding apparatus mounted on the two end frames was capable of measuring the temporal profile of the plate at the midsection. This profile was measured in the unloaded condition (reference state) and at various stages during loading. The difference between the height of the plate at the deformed and the undeformed configuration would give the lateral deformation of the object due to wind pressure and wind suction.

Fig. 8 shows the result of continuous profile measurement in the 6-meter direction. This Figure shows the mid section profile of the sandwich structure at the reference state and at various internal pressure loading up to 49 mbar. From this diagram, a lateral deformation of the structure at the apex of the dome at the wind suction of 45 mbar a value of about 110 mm can be calculated.

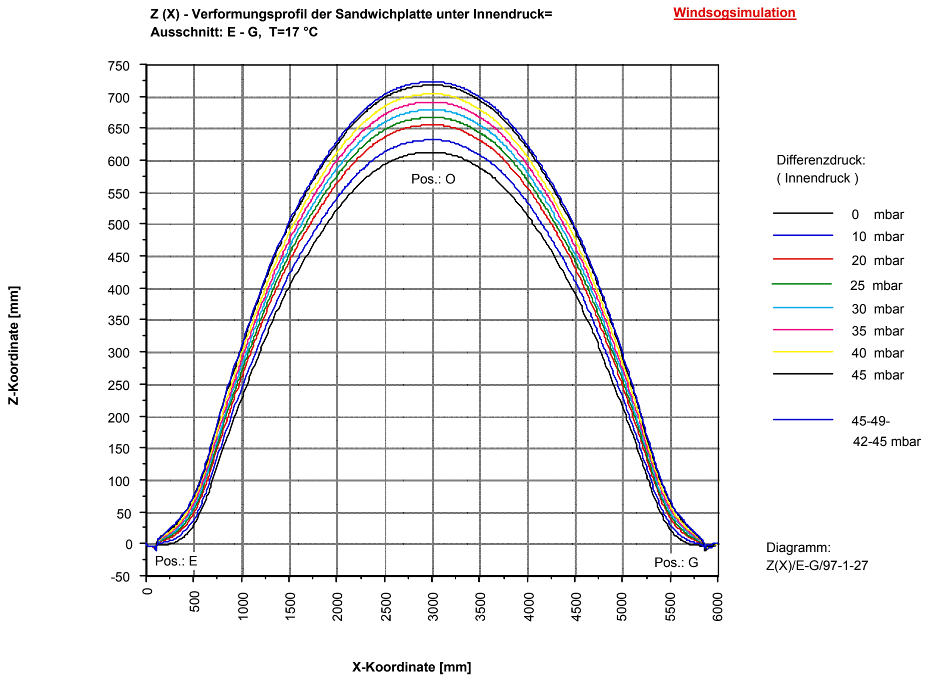


Fig. 8: Undeformed and deformed mid section profile of the sandwich structure in the x direction at various values of wind suction

Paper published in: Structural Engineering International, No. 4, pp292-297, 1999.
Large Prefabricated Thin Faced Sandwich Elements for Antenna coverage
M. Farshad and P. Flüeler

5.3.2. Strains

Table 2 gives the results of strain gauge measurements at maximum wind pressure and wind suction loading.

Table 2: Result of strain gauge readings at maximum wind suction loading

Position of strain gauge	Strain values [%]		
	Wind suction simulation (internal pressure) at 35.165 mbar	Wind suction simulation (internal pressure) at 49.472 mbar	Wind pressure simulation (internal vacuum) at 33.6 mbar
1	0.342	0.469	-0.240
2	0.492	0.669	-0.335
3	0.317	0.419	-0.305
4	0.464	0.640	-0.320
5	0.095	0.130	0.020
6	0.119	0.158	0.070
7	0.064	0.086	-0.050
8	0.052	0.072	-0.022
9	-0.091	-0.108	0.040
10	-0.238	-0.334	0.125
11	-0.307	-0.411	0.210
12	-0.253	-0.333	0.090
13	-0.048	-0.066	-0.040
14	-0.051	-0.073	-0.050
15	-0.067	-0.065	-0.240
16	0.268	0.406	-0.160
17	-0.287	-0.383	-0.310

5.3.3. Results of acoustic emission (AE) monitoring

As mentioned before, the main goal of acoustic emission monitoring of the plate behavior during wind pressure and wind suction simulations was to analyze the events related to energy release due to local failure and hence to be able to predict the damage of the structure. This prediction was necessary since the spontaneous escape of a relatively high quantity (about 30 m3) stored pressurized air underneath the structure could jeopardize the safety of the persons and the equipment.

The acoustic emission monitoring was carried out on the basis of the guidelines set by the "Committee for Acoustic Emission from Reinforced Plastics" (CARP) [2]. According to this guideline, a loading program composed of stepwise loading and unloading was to be followed. The level of intensity of the acoustic emission activity would then determine the incipient damage in the structure.

In wind suction (internal pressure) tests, the AE activity was observed even at relative lower value of loading; the AE activity increased by the increase of the internal pressure. By constant load of 20 mbar, remarkable AE activity (> 55 $dB_{ref1\mu V}$) occurred which lasted more than two minutes. By reloading up to 33 mbar and further to 35 mbar, the AE activity was registered by the majority of the sensors. This was attributed to the corner construction used.

The AE activities were smaller in the wind pressure simulation (internal vacuum) tests than the wind suction simulation tests. Most of the AE occurred at the inside surface of the structure. Only by the loading level of –24 mbar a higher AE amplitude (> 65 dB) was registered.

5.3.4. Natural frequency of the structure

The first three natural frequencies of the sandwich structure with real boundary conditions were obtained by the ambient vibration method. The excitation from slamming of the doors and the operating machinery was enough to bring the structure into its natural vibration. The ambient vibration measurement gave the values 13.9 Hz, 18.4 Hz, and 25.8 Hz for the three natural frequencies of the structure.

6. THEORETICAL MODELING AND FINITE ELEMENT SIMULATION

6.1. Theoretical correlation of wrinkling results

For theoretical correlation of the wrinkling experiments, analytical modeling and Finite Element simulation were used. There are different analytical models available for the evaluation of the critical force P_{crit} to induce wrinkling. A number of these models are derived from the stress state in a plate on an elastic foundation. In these models, the facing corresponds to the plate and the core acts as the elastic foundation. For sandwiches with thin facings one may assume that the individual facings have no interaction and that each facing reacts independently with its immediate bedding.
One of the theories dealing with the face wrinkling is based on the model of the plates (facing) on elastic foundation (core). This theory gives an estimation for the critical wrinkling force for the facing wrinkling in a sandwich panel subjected to the in-plane compression. The critical wrinkling load, per unit width of the compressed panel, given by this theory reads as follows:

$$P_{crit} = \kappa \cdot t_f \cdot \sqrt[3]{E_f \cdot E_c \cdot G_c} \quad (1)$$

In this formula, Pcrit is the critical wrinkling load per unit width of the panel, Ef is the Young's modulus of the facing, tf is the thickness of the facing, and Ec and Gc are the Young's modulus and the shear modulus of the core, respectively. The theoretical value for the factor κ is: $\kappa = 1.7$. Based on the experimental investigations, a design value of κ = 0.91 is recommended
The formula (1) gave a theoretical wrinkling load of 23500 N in the 0°/90° direction and a 16200 in the 45° direction. These values should be correlated to the corresponding experimental results of 17000 N and 16000 N, respectively. The material parameters, which were assummed in the theoretical calculation of wrinkling load were: Facing: E = 20700 N/mm^2, Core: E = 33 N/mm^2, G = 15 N/mm^2.
The theoretical predictions of the wrinkling load based on the equation (1) are compared with the experimental values of the wrinkling in table 5.

Table 3 summarizes the results of wrinkling tests on various compressed sandwich panels and the related compression failure. It also gives the theoretical estimation for the wrinkling load.

Table 3: Theoretical and experimental values for wrinkling load of compressed sandwich panel

Sample No	Failure mode*	Test [N]	Theory N]
1	B	23114	30717
2	B	31026	32046
3	C/W	30650	38392
4	B	21659	18838
5	B	15653	11903
6	C/W	13700	28822
10	C/W	42000	27721
11	C/W	29000	31189
12	B	14826	18320
13	C/W	23375	19901
14	B/W	54274	31793
15	C/W	32980	32558
16	C/W	16665	15526
17	C/W	29500	32046

- B: wrinkling and compression failure at the panel ends
- C/W: local wrinkling and compression failure
- B/W: wrinkling in a zone

The wrinkling behavior of the sandwich samples was also simulated by the Finite Element method. The Finite Element software used was COSMOS/M version 1.75. A non-uniform meshing was made and a linear eigenvalue buckling analysis with a displacement-controlled loading was performed. Fig. 9 shows the wrinkling pattern in a sandwich panel under axial compression. The calculated critical displacement obtained from the linear eigenvalue buckling analysis was 1.48 mm; resulting in a critical axial strain of 0.7%; this should be compared with the experimental value of 1%.

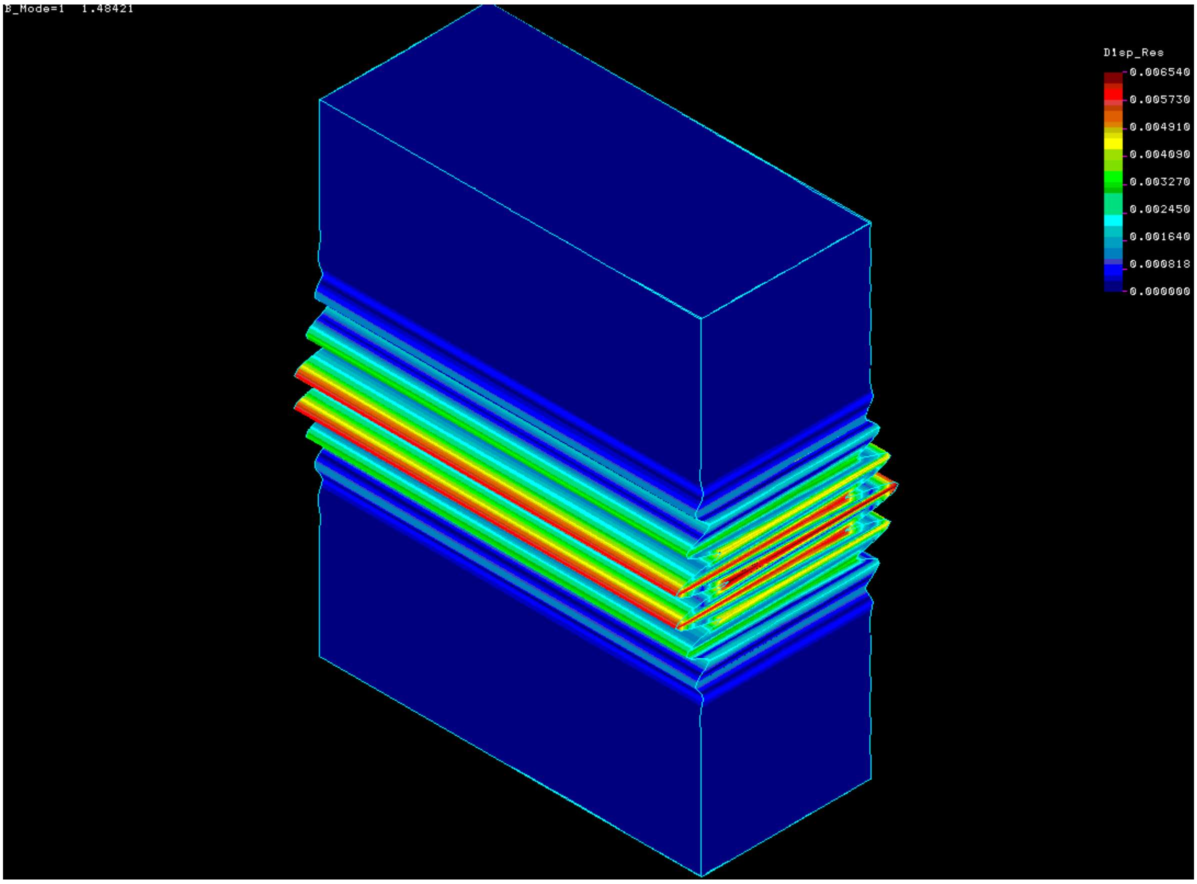


Fig. 9 Wrinkling pattern in a compressed sandwich panel

6.2. Finite-Element-Analysis of Sandwich Structure

In order to plan the tests on the whole structure, to correlate the experimental results, and to verify the adequacy of structural strength and stiffness, a set of Finite Element simulations of the structure for the wind pressure and wind suction were performed. Additional FE analyses included simulation of the structural behavior under thermal gradients and buckling analysis of the shell structure.
The geometry and the boundary conditions used in the FE analysis corresponded to those used in the construction site. The assumed properties of the structure were:

Elasticity moduli of the facing: E_f = 9512 N/mm^2, ν_f = 0.3

Elasticity moduli of the core: E_C = 45 N/mm^2, ν_C = 0.3

Thickness of the core: t_c = 100 mm, thickness of the top facing: t_1 = 0.66 mm, thickness of the bottom facing: t_2 = 0.44 mm

Length of the element: a = 6600 mm, width of the element: b = 6000 mm, rise of the middle shell: f = 600 mm, basis radius of the spherical shell: R = 2500 mm

For FE simulation of the structure under wind suction, a uniform internal pressure of p = 0.0049 N/mm^2 = 490 kg/m^2 was applied on the system. For wind pressure simulation, an internal vacuum of p = 0.0049 N/mm^2 = 490 kg/m^2 was applied. A total of 2764 sandwich shell elements were used and a linear elastic and static analysis was performed.
Fig.10 depicts the FE mesh; Fig. 11 and Fig. 12 show some results of FE analysis for the wind suction loading. Fig. 12 shows the resultant displacement field and Fig. 12 presents the von Mises stress field in the structure. As we note, the maximum deflection does not occur at the middle of the element, but it occurs at the transition zone between the dome and the plate. This prediction was also experimentally verified. Fig. 11 gives a calculated value of 116 mm for the deflection of the sandwich element under wind suction of 0.049. This result was also experimentally verified (see table 4)

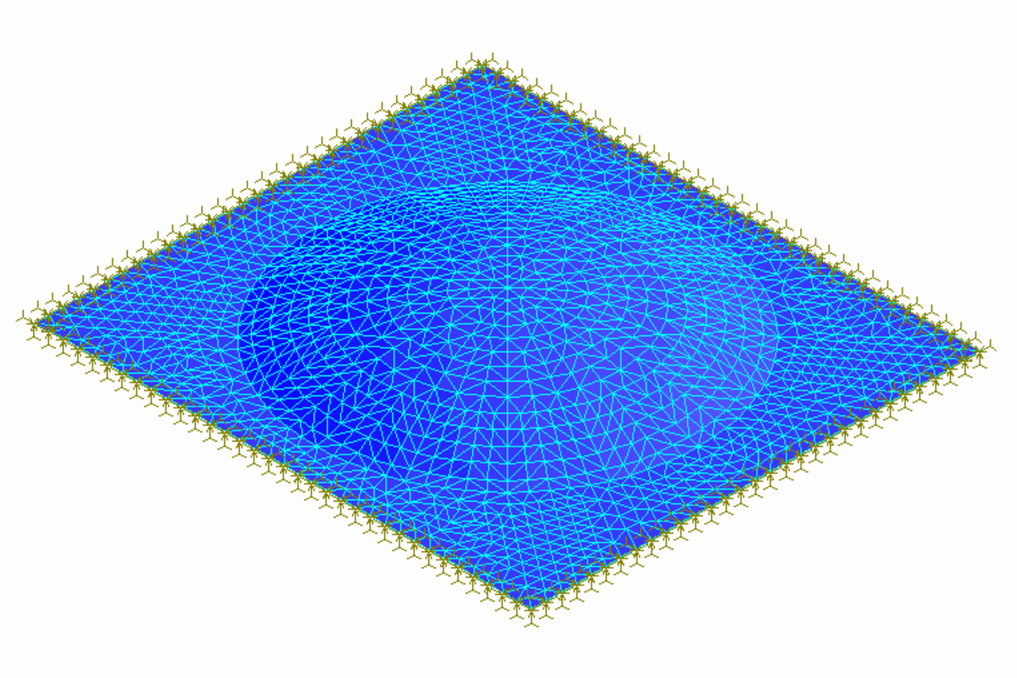


Fig. 10: Finite-Element model of 6.0 m x 6.6 m sandwich element

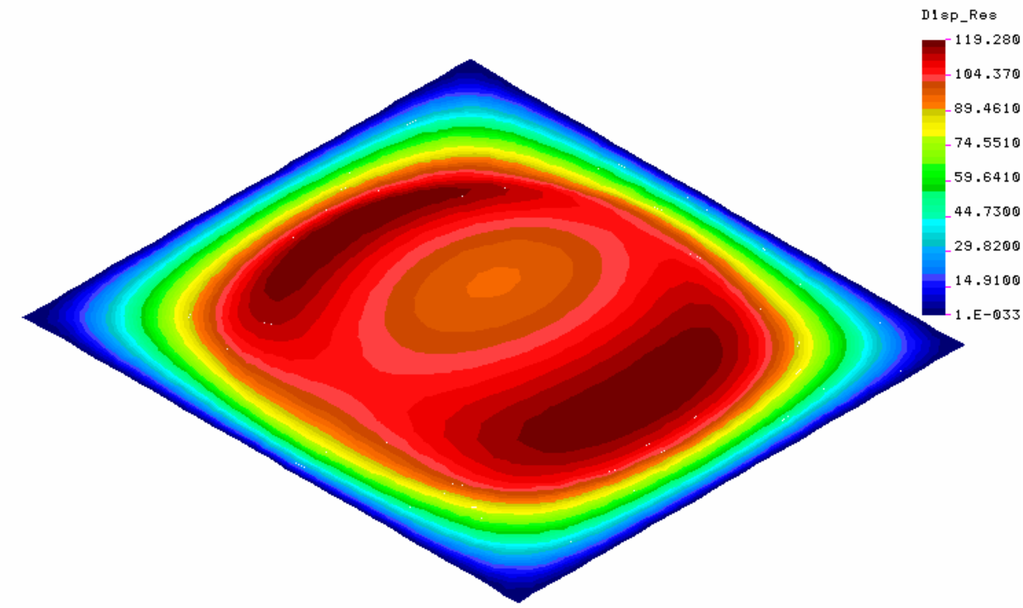


Fig. 11: Resultant displacement field in the sandwich structure under wind suction (p = 0.049 mbar)

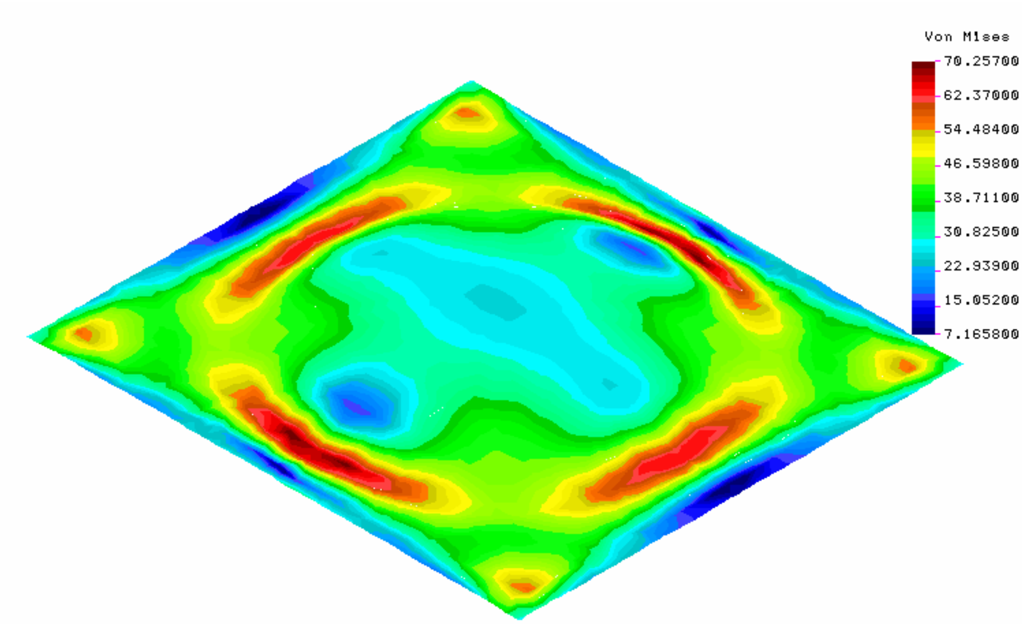


Fig. 12: Von Mises stress field in the sandwich element under wind suction (p = 0.049 mbar)

Tables 4 and 5 summarize the results of lateral deformation measurements as well as the values calculated by the Finite Element analysis at some of the key points of the sandwich structure. Table 4 shows that the maximum lateral deformation occurs not on the apex of the dome, but somewhere on the transition zone between the dome and the flat plate. This conclusion is both theoretically and experimentally verified.

Table 4: Selected values of measure lateral deflection of the sandwich element in wind suction experiments (up to an internal pressure of p = 0.0049 N/mm^2)

Position on the sandwich element	Lateral displacement, Z direction [mm]			
	6 meter direction (X)		6.6 meter direction (Y)	
	measured	calculated	measured	calculated
Apex of the dome shell (Point O)	111	95	111	95
Transition zone between the dome and flat plate (1)	125	119	116	105

(1) Transition zone between the dome and the flat plate located about 1650 mm from the apex of the dome

Table 5: Selected values of measure lateral deflection of the sandwich element in wind pressure experiments (up to an internal vacuum of p = -0.00336 N/mm^2)

Position on the element	Lateral displacement, Z direction [mm]			
	6 meter direction (X)		6.6 meter direction (Y)	
	measured	calculated	measured	calculated
Apex of the dome shell (Point O)	60	82	60	82
Transition zone between the dome a flat plate (1)	50	66	63	72

(1) Transition zone between the dome and the flat plate located about 1650 mm from the apex of the dome

7. PRODUCTION, TRANSPORT, AND MONTAGE OF PREFABRICATED ELEMENTS

Realization of a 40 m^2 large prefabricated sandwich element with very thin facings, its transport to relatively far distance up to the construction site on top of mount Säntis, and its montage on the steel construction in an inaccessible place was a challenging task. It was required that a relatively perfect structural element free from imperfections was to be produced and with no damage transported and installed. To meet this challenge, a thorough program including production, transport, and installation phases was fulfilled. Continuous supervision and quality assurance accompanied these realization phases. This section contains a brief description of these activities.

7.1. Production of prefabricated sandwich element

Production of 6m x 6.6-m sandwich element with thin facings was realized through several stages. These included:

(1) Production of 100 mm thick integral PEI foam
(2) Division of the element into 30 pieces
(3) Cutting of the foam panel into flat pieces with given goemetries, such that upon foaming of pieces into curved elements the final shape of pieces would be obtained.
(4) Thermal shaping of flat pieces of the foam into curved shapes
(5) Transport of the foam pieces to the production place
(6) Assemblage of the foam pieces on a prefabricated wooden frame and gluing them with epoxy matrix
(7) Placement of GFRP plies on the finished foam element; these plies had been impregnated with the epoxy matrix and were rolled on a preparation table. Upon unrolling, the matrix-wetted plies were placed on the surface of the element. Prior to placing of the GFRP strips, both surface of the foam element was treated with Epoxy 545 primer. Each facing consisted of 2 layer of GFRP plies of the type US Glass 299 g/m^2 which were laid at 0°/90° orientation. The facings were laminated with SP Ampreg 20 Epoxy matrix in a vacuum process and were consequently treated thermally. Finally, a PUR coating of the type IMRON 700 was applied to the outer surface of the element. In total, 18 elements of this geometry were produced. There were also other 34 sandwich elements for the antenna coverage with other geometries, which were manufactured by the same process. The latter element types were used for corners, borders, and the edges of the antenna coverage system.

7.2. Transport

Transport of a 40 m2 large prefabricated sandwich elements with very thin facings from several hundred kilometers from fabrication place to the mountain base and from there to the construction site located at top of 2500 meter over the see level produced many practical challenges. The 6 m x 6.6 m-prefabricated elements were transported with the ground transportation to the base station of mount Säntis; the Schwägalp; from there, the elements were transported with the cable car with a special hanger construction to the top of the mountain. Local transport of the elements and their placement on the steel construction took place with a crane system on the top of the building. Fig. 13 shows an element which is being lifted by the crane and is being displaced to the steel framework on the extended side of the building.

Fig. 13: Local transport of a sandwich element from the building roof to the steel framework at the side of the building

7.3. Montage

The prefabricated sandwich elements were mounted on the structural steel construction, which was constructed as extension to the existing building. Fig. 14 shows a stage of mounting of elements on the steel structure. The elements were fixed on the steel frame by means of a row of bolts, which were placed about 300 mm apart from each other. The fixation took place by fastening of a series of bolts on steel strips placed on the U profiles located around the element. To assure the water tightness, a 300-mm wide cylindrically formed GFRP shell with integrated elastomeric profile bridged the connection line between two elements. Fig. 15 shows a partial view of the outer surface of the antenna coverage; it shows the row of bolts and the watertight curved GFRP profile.

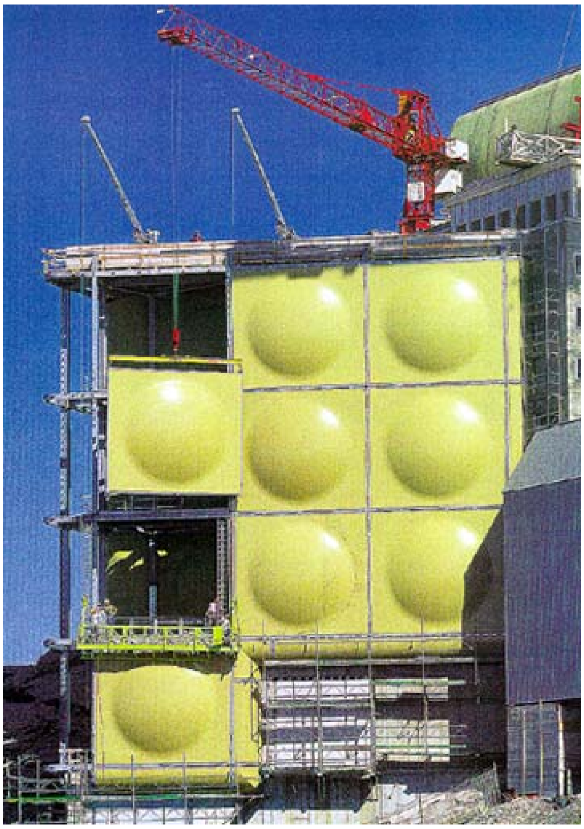


Fig. 14: Montage of a sandwich element on the steel constriction

Fig. 15: Detail of the structural connection of sandwich elements on the steel construction. This Figure shows the crossing of bolted GFRP cover profiles, which are used to provide watertightness for the antenna coverage system.

7.4 Supervisory work

The engineering realization of a multifunctional sandwich structure made of large prefabricated thin-faced elements was a very delicate project, which would require serious planning and continuous supervision at all stages from conception to acceptance. The supervisory work included comparisons between conceptions and reality, between structural design and structural performance, and quality assessment of production of sandwich elements at various stages, including transport, and montage. This supervision should, principally, continue after installation of elements and throughout the service life of the structure. The ambient vibration measurements and the related theoretical correlation, described before, were aimed at the periodical monitoring of the structural response in comparison with a reference state. Figure 16 shows the construction site and the existing antenna coverage at the southwest side. Figure 17 shows the south west view of the new antenna coverage system.

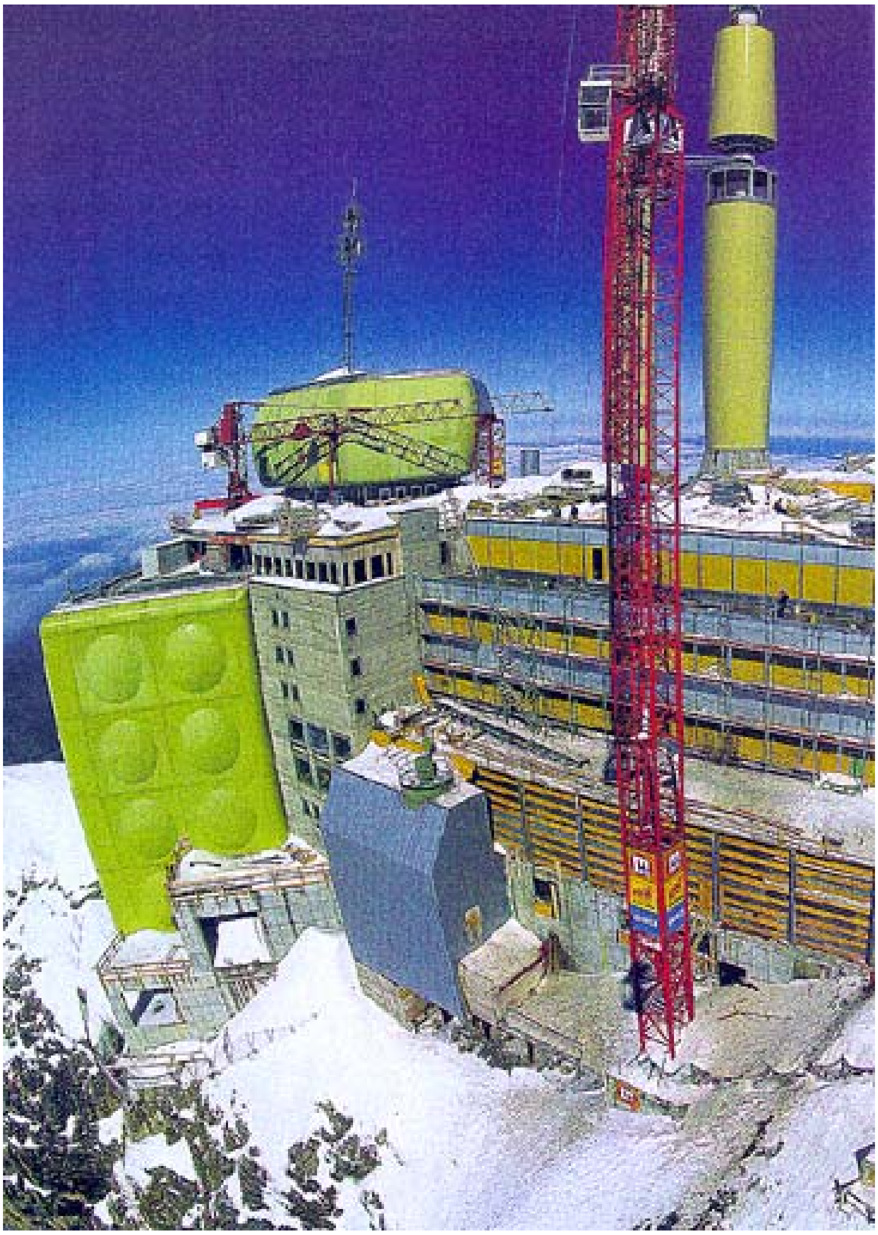


Fig. 16: Construction site on mount Säntis. This Figure shows the southwest view of the existing old antenna coverage. The building itself was to be extended and the existing cladding was later to be replaced by new elements.

Fig. 17: Southwest view of the new antenna installations on mount Säntis. This Figure shows the new 6 m x 6.6 meter new sandwich elements and the other sandwich border elements, which together constitute the antenna coverage at the southwest side.

8. CONCLUDING REMARKS

In this article, aspects of a non-conventional engineering project from conception to full realization was reported. Such project with many fine details ranging from requirements, engineering calculations, material and system tests, production, transport, and montage could only be successfully realized if a systematic process and a holistic approach was adopted. One of the delicate details of the structure involved the design and production of sandwich elements with extremely thin faces. These elements

should be structurally effective, radioelectrically efficient, and practically producible. Any imperfection in the shape and in the sandwich construction could not be tolerated. Moreover, no damage was to occur in the transport and the montage of the sandwich elements. The whole antenna coverage system was to be watertight and was to function for many years. During its service life, harsh winters accompanied by snow, ice, and winds with very high speeds up to 240km/h were expected to occur. The thin faced sandwich elements on the new antenna installations on the mount Säntis were realized to meet these challenges. This part of the project was realized in 1998.
In this project, many companies and a chain of specialists were involved; who contributed to the success of the project. The role of EMPA was to accompany the project from its conception to realization; to set up the requirement lists, to offer consulting on the choice of material and system, to test material samples and structural element, to check the engineering calculations and dimensioning, to supervise various stages of production and montage.

AKNOWLEDGEMENTS

Technical support of specialists in the division of Polymers/Composites and also other divisions of EMPA and also the close collaboration of companies and Organizations involved in the planning and the realization of the project is thankfully acknowledged.