THE CFRP SANDWICH PANEL FOR AIRCRAFT NOSE STRUCTURE

Y.Hirose*, K.Kosugi*, M.Nishitani*, H.Sashikuma*, M.Imuta*, H.Fukagawa*, H.Kikukawa**
*Kawasaki Heavy Industries, Ltd.,
1 kawasaki-cho, Kakamigahara City, Gifu-Pref., 504-8710 Japan
**Japan Aircraft Development Corporation,
Toranomon Daiichi BLDG., 2-8 Toranomon 1-chome, Minato-ku, Tokyo, 105-0001 Japan

Keywords:*CFRP,Nose structure,Sandwich Pnael,Design,Fabrication*

Abstract

Five-year research activities have been started in 1999 sponsored by the New Energy and Industrial Technology Development Organization (NEDO). The objectives of the research are to apply new technologies to the nose structures of the commercial aircraft. The CFRP sandwich panel is selected as the one of them. The design, data acquisition tests and the fabrication of the CFRP sandwich panel subcomponent article were conducted.
Among these activities, the FEM analysis and stress analysis were conducted supported by the tests. In order to obtain the design data and to back up the structural design, structural element test of the panel joint and core splice were conducted. The bird impact tests of the inclined flat panels were also conducted. Through these tests, CFRP sandwich panel with the surface plate of 1.8 mm thickness could sustain the impact of 4 lb bird successfully. Intensive analysis was also performed to the joint design of sandwich panel, especially the periphery of the open section of windshields. Based on this analysis, flush butt joint concept was selected. Through these design, 23% weight reduction and 98% parts count reduction for composite parts is estimated. This result meets our target. Full-scale test will be conducted in 2003.

1 Introduction

The nose of transport category aircraft is a tri-dimensional complicated built-up structure. Though there is much room to reduce fabrication cost, few efforts have been conducted. This time a five (5) year research activity has been started in 1999 sponsored by NEDO and Japan Aircraft Development Corporation (JADC).

Through this research, the co-cured Carbon Fiber Reinforced Plastic (CFRP) sandwich panel technology is to be developed and applied to the aircraft structure in order to reduce its structural weight and parts count.

Conventional honeycomb core sandwich panel has the disadvantage of water entrapment. In order to overcome this difficulty, synthetic core sandwich panel is selected.

The final goal is to achieve a 20% weight reduction and an 80% parts count reduction through these applications.

This paper describes the introduction of the development plan and the interim results.

2. Development Plan

The development plan consists of two phases namely “the basic technology establishment phase (1999-2001)" and “the structure engineering verification phase (2002-2003)”.

In the basic technology establishment phase, a preliminary design, material characterization tests and process tests will be conducted in order to access engineering application feasibility.

In the structural engineering verification phase, design and verification tests of a full-scale nose structure will be performed in order to show compliance with the Federal Aviation Regulation (FAR) Part 25.

This paper describes the interim results obtained in the basic technology establishment phase. (1999-2001). [1]

3 Structural design and analysis

3.1 Conceptual Design

Based on the survey of existing aircraft structures and considering the characteristics of this new technology, the nose skin panels has been selected as a candidate.

The conventional nose structure is comprised of skin panels, sheet metals, stringers and frames. In order to reduce weight and parts count, it has been found that a simple material replacement of these individual parts was not enough. To achieve these goals, it was necessary to replace all the parts by a single structure with sufficient rigidity and strength. A CFRP sandwich panel is suitable in this case. This panel consists of 180 centigrade cured CFRP surface skins and foam core. [1] The structural concept is shown in Fig.1.

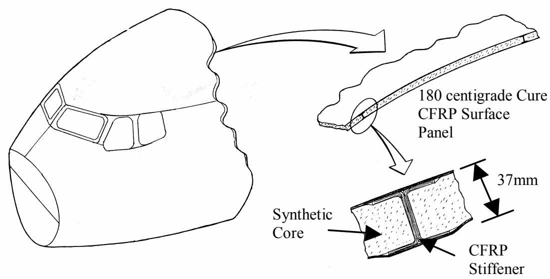

Fig.1 Structural Detail of CFRP Sandwich panel

3.2 Detail design

Detail design was carried out supported by analysis. The conventional metal skins were replaced by the synthetic core sandwich panels. The nose structure skin panel consists of an upper sandwich panel and a lower sandwich panel. Each sandwich panel was divided into two parts, right hand-side and left hand-side for the production efficiency, since one piece large panel is expected to have more problem in production, such as difficulty in the lay-up process. These panels are jointed with flush butt splice backed up by a metal beam. The joint concept was selected due to an easy visual inspection.

A Cu expand metal is installed to the outer surface of the sandwich panel for a lightning protection because the nose portion of an airplane is defined as a zone 1 for a lightning strike. Metallic longitudinal beams for the panel splice are used as the current flow path in case of the lightning strike. The effect of a thermal stress was also investigated and no significant effect was confirmed.

The existing pressure floor panel support structure is a built up sheet metal component and has many open holes for control cables and piping. Therefore, it is not suitable for the application of composite materials. It has been found that the precision casting technology is suitable for such a complex one-piece large structure. [2]

The conventional forward pressure bulkhead is a built up of several components and has to maintain structural integrity against rapid decompression. The casting structure is not suitable due to its low ductility and a composite is also not suitable due to many open holes that are required for systems installation. In order to reduce its part count, an application of the Friction Stir Welding (FSW) with the integral panel is expected to be suitable. [2]

There may be some candidate structures for the application of these technologies, i.e., nose landing gear well and the rear bulkhead, etc. As for these structures, the feasibility of the application will be surveyed as the research progresses.

The detail design of the nose structure is shown in Fig.2.

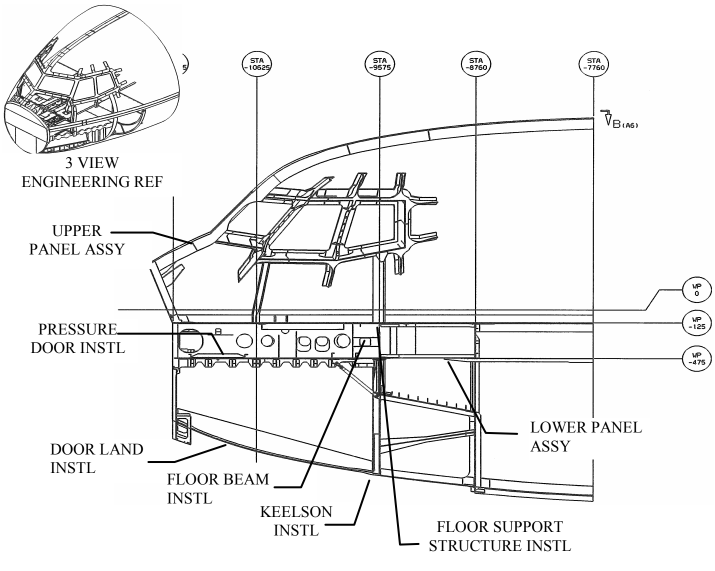

Fig.2 Detail design of the nose structure

The detail design of the upper sandwich panel is shown in Fig.3.

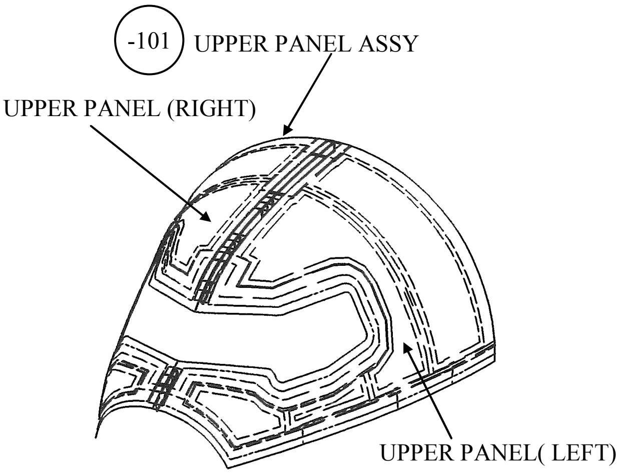

Fig.3 Detail design of upper sandwich panel

The panel joint is one of the critical areas in the nose structure. Special attention must be given to its design, fabrication and assembly.
A weight, strength, fabrication and assembly processes were assessed for several candidate joint concepts. A flush butt splice joint concept was selected from the inspection and assembly point of view. This type of joint can be inspected from both inner and outer side, which is an important aspect for the aircraft inspection. As for the assembly, flush butt joint is easy to assemble due to the adjustment by the shim.
The flush butt splice is shown in Fig.4.

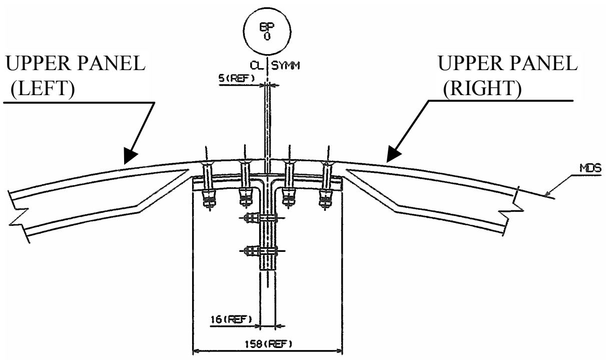

Fig.4 Detail design of the panel joint

3.3 FEM analysis

Considering the above concepts, the detail design of the nose structure was carried out and the Finite Element Method (FEM) analysis was conducted for the critical loading conditions of a pressurization load and landing loads. As for the FEM model, the core, the outer and inner skins of the sandwich panel were modeled separately in order to achieve better results. (See Fig.5) Through this analysis, internal loads for each part were obtained and pad-up areas were defined accurately. It was confirmed that the pressurization load was sustained by the CFRP sandwich panels and the local bending moment due to open sections like a windshield cutout was also sustained by the rigidity of the sandwich panel.

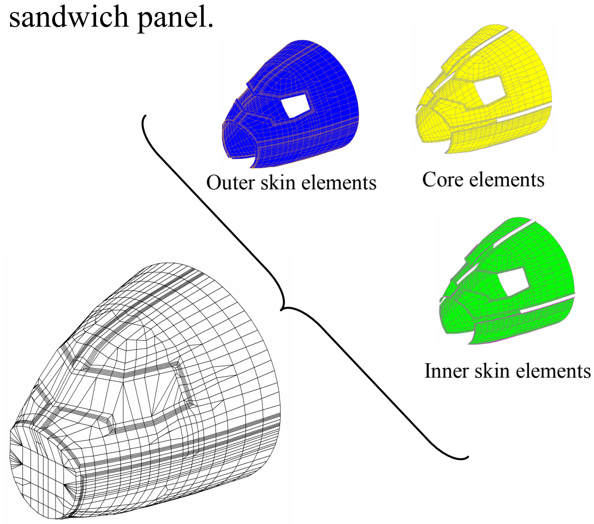

Fig.5 FEM Model concept

The result of the FEM analysis is shown in Fig.6. Through this result, it has been established the required ply thickness of the outer and inner skin to be 2 plies except for the local pad-up.

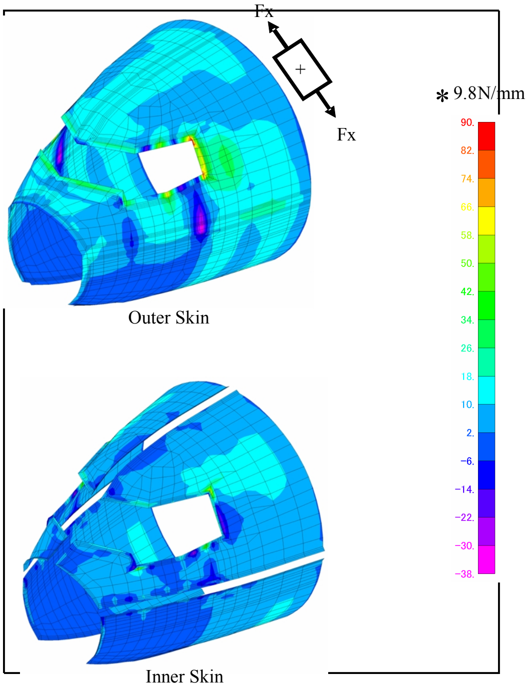

Fig.6 FEM analysis Result

3.4 Joint analysis

Adding to the nose structural analysis, some detail analyses were necessary. Among them, the panel joint analysis was the most critical. The load distribution between inner and outer skins was analyzed.

The detail FEM model of the panel joint was made and the design allowable of the joint was obtained through the structural element tests (See 4.1.2)

The detail FEM model is shown in Fig.7. In this model, individual layers of CFRP laminate were modeled and strain distribution of each layer was obtained in order to decide the optimum thickness.

The strain distribution is shown in Fig. 8.

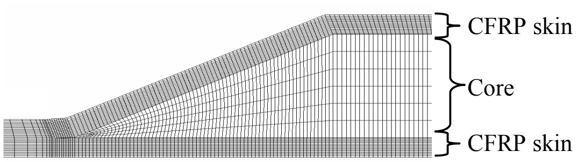

Fig.7 Detail FEM Model of Panel Joint

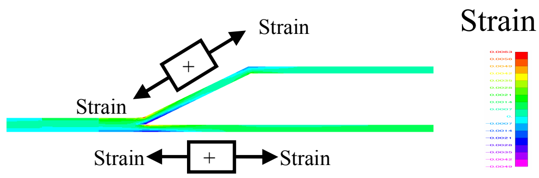

Fig.8 Strain Distribution of Panel Joint

Based on the strain distribution, the optimum thickness of 24 plies was determined for panel joint and surface plate thickness of 2 plies for sandwich panel skin has also been selected.

3.5 Stress analysis of nose structure

Based on the internal loads obtained by the FEM analysis, stress analyses were carried out. The pressurization load was the critical load for the sandwich panel. A nose landing gear spin up load of a landing load is the critical load for the nose landing gear well cutout. The minimum thickness of outer and lower skins was confirmed due to the damage tolerance consideration.

The analyses of all parts of the nose structure had enough margin of safety and the minimum figure was 0.04 at the connecting point of sandwich panels. (See Fig.9)

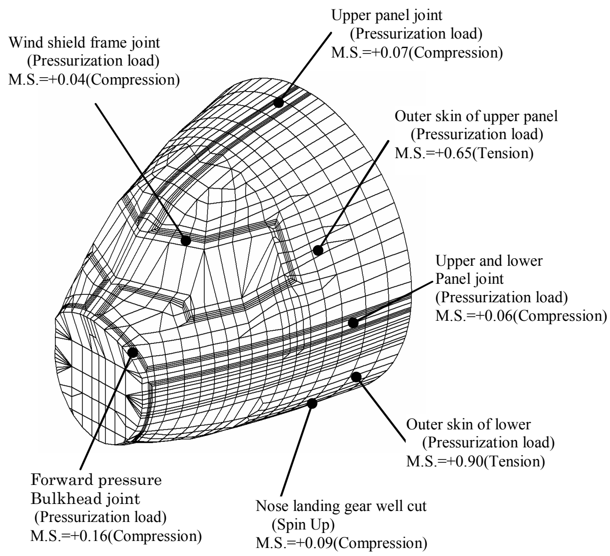

Fig. 9 Margin of safety for nose structure

Based on the detail design, 23% weight reduction and 98% part count reduction was estimated for the composite parts. This result meets our target.

4 Experimental results

4.1 Design data acquisition tests

4.1.1 Core splice test

The core of the sandwich panel is comprised of several small cores of 1m by 1m. Each core is spliced with a foam tape adhesive. Therefore, the strength of the core splice portion was obtained through the tests. The four point bending tests of the core splice parts were conducted as supporting data.

The test specimen is shown in Fig.10.
Two types of specimens such as with a spice and without a splice were tested and compared.

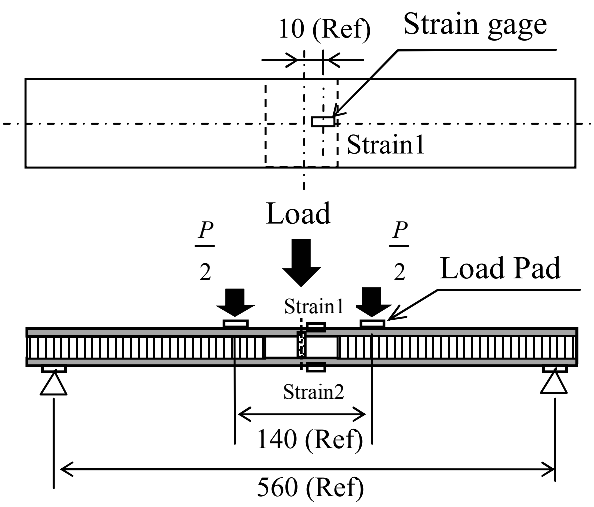

Fig.10 Core splice test specimen

The test situation is shown in Fig.11.
The test results were shown in Table 1.
The table shows that the core strength of both types of splices has no significant difference. Therefore, it has been confirmed that the core splice does not lead to the core strength degradation.

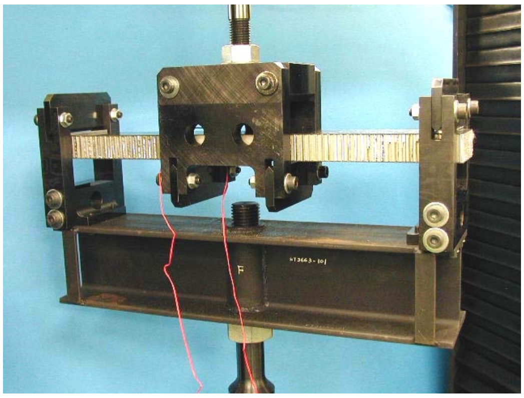

Fig. 11 Core splice test situation

Table 1 Core splice test results

	Specimen	Average strength	Remarks
Compression strength(MPa)	With a splice	**143**	
	Without a splice	**149**	

4.1.2 Structural element test for panel splice joint

Relating to the sandwich panel joint, the internal strain distributions were obtained through the FEM analysis. In order to estimate the joint strength, it was necessary to confirm a fracture mode and fracture strain of the joint. Therefore, the structural element tests of the panel joint were conducted.
The test specimen simulated the longitudinal panel joint and test load simulated the pressurization load that was the critical load for the joint.
The test status of the joint portion is shown in Fig.12.

The length of the specimen is 0.7m, which has enough length to eliminate an inadequate effect of the test load. The static test and fatigue test were conducted.

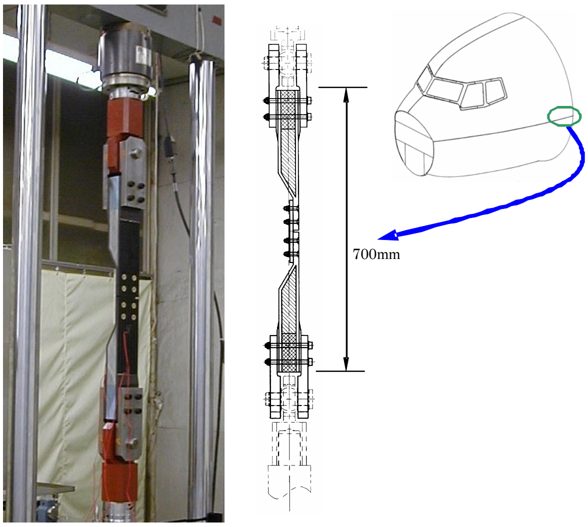

Fig.12 Test status of the panel joint test

The test results are shown below.

1. An initial crack was occurred at the edge of the core and average load in case of the initial crack occurrence was 39kN.
2. Final fracture was occurred at fastener joint and average fracture load was 112kN.

Through these results, it has been confirmed that the joint has an enough margin from the initial crack occurrence to the failure

The design improvement to minimize this margin will lead to the further weight reduction.

The strain – load diagram is shown in Fig.13.In this diagram, the occurrence of the initial crack indicated an abrupt change of the curve in the strain gage C.

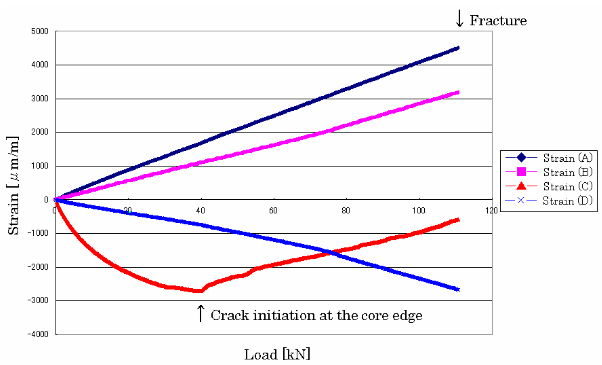

Fig.13 The strain – load diagram of Joint

The crack initiation point is shown in Fig.14.

The abrupt change of the stiffness has caused the initial crack.

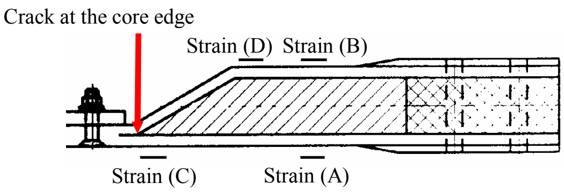

Fig.14 Crack initiation point

4.1.3 Hail attack test

The static and fatigue tests of damaged specimens due to the hail attack have been conducted, and it was possible to obtain the design data.

The relation between energy level and the hail attack damage is shown in Fig.15.

No significant damage could be detected with a visual inspection. Through these tests, the damage characteristics of the CFRP sandwich panel have been obtained.

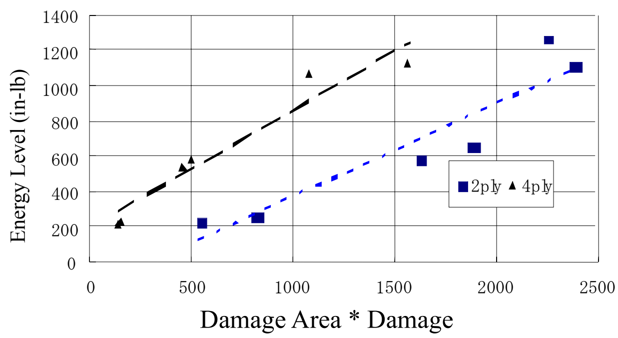

Fig.15 Damage Characteristic Test Results

The static test data is shown in Table 2.

Table 2 Static Test Results

	Average Value (N=3)	Standard Deviation	Cv(%)
Damage A	**126** MPa	1.54	1.23
Damage B	**125** MPa	4.58	3.66
Flaw	**153** MPa	2.28	1.49

Damage A : 2.5Dia inch Hail attack Damage.

Damage B : Tool Damage of 0.1in depth.

Flaw : 0.75in *0.75in flaw

These fracture loads were much higher than those of applied load during the operation. Therefore, no significant effect of the hail attack damage was confirmed.

The fatigue test data were shown in Fig.16.

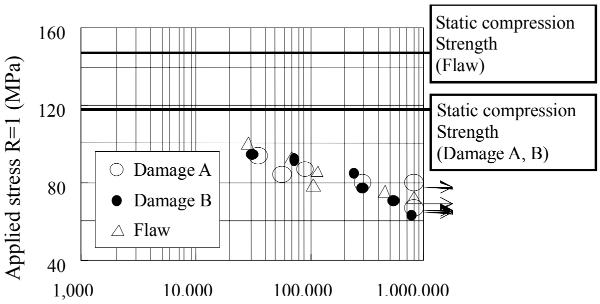

Fig.16 Fatigue Test Result

.

Through these test, the endurance of more than 10^6 was confirmed for the half of the fracture load. Though the final endurance characteristic should be estimated for the actual load spectrum, the prospect of the damage endurance was obtained. [3]

4.1.4 Bird impact test

In order to obtain the design data for bird strike damage and later to validate the analytical method, bird strike tests of flat CFRP sandwich panels has been conducted.

Two types of sandwich panel test articles were selected and tested. One had 4ply $[(45,-45)/(0,90)]_2$ surface skin and the other had 8 ply $[(45,-45)/(0,90)]_4$ surface skin.

The tests have shown that both articles could sustain 4-pound dummy bird impact at 320 kts velocity.

The test article with 4-ply skin after the impact is shown in Fig.17 and the dent distribution due to the impact is shown in Fig.18.

Based on these data, 4-ply skin has been selected for the critical areas against the bird strike.

The boundary condition of the test article showed a significant effect on the damage characteristics. This information was valuable to validate and adjust the FEM model. By comparing the test data with the analytical model, the effect on the boundary condition of the test article will be estimated and the strength against bird strike will be predicted. [3]

Fig.17 Damaged test article

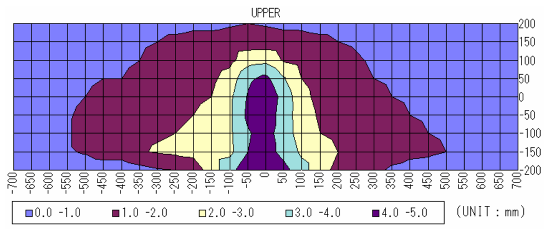

Fig.18 Dent distribution

4.2 Manufacturing concept

Since the application of a foam core sandwich panel to the nose skin panel for a commercial transport is a new technology, it is expected to have many manufacturing challenges ahead, such as low cost fabrication and a development of the processes to obtain low distortion when manufacturing sizeable structures.

4.2.1 Core molding concept

Cores of sandwich panel were divided into several small cores which are spliced each other. For instance, the left-hand side of the upper sandwich panel consisted of 10 small cores. Each core had a different size and a curvature. For a conventional process, an individual molding jig, so called solid type jig, was necessary for each core. In order to achieve the low cost fabrication, it was necessary that cores with the different shape could be molded by a single device as much as possible. Our idea of a multi-support molding method was one of such devices. The concept of the multi-support molding jig is shown in Fig.19. In this device, heights of many multi-support bars were controlled with a computer in order to give an appropriate shape of each core.

The flat core was heated in an oven and transferred to the multi-support jig. The transferred core was put on the multi-support bars and pressed with a vacuum pressure to be molded.

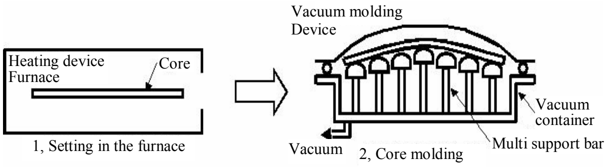

Fig.19 Multi-support molding concept

Through many trials in a core fabrication, the manufacturing data, such as core spring-back data, were obtained and a shape of the multi-support head was decided. These data were applied to the sandwich panel fabrication.

The multi-support core-molding jig to be used in the sandwich panel fabrication is shown in Fig.20

Fig. 20 Multi support core molding jig

Although a current device could not be applied to the core with an abrupt changed in curvature, it was very effective for the relatively simple contoured one.

4.2.2 Sandwich panel molding concept

The development of the manufacturing process to eliminate the distortion of a sizeable panel has been a critical topic of the research.

In order to solve this problem, the following ideas were applied to the molding jig.

1. The same CFRP material as the skin panel was applied to the jig material to minimize the effects of the differences in the thermal expansion.
2. A uniform temperature distribution of sandwich panel in a cure process can be achieved due to an excellent thermal conductivity of the CFRP.
3. To maintain a uniform heating, ventilation holes were allocated appropriately in the molding jig.

The CFRP sandwich panel molding jig is shown in Fig.21.

Fig. 21 sandwich panel molding jig

The molding process of the sandwich panel for a tool try article is shown in Fig.22.

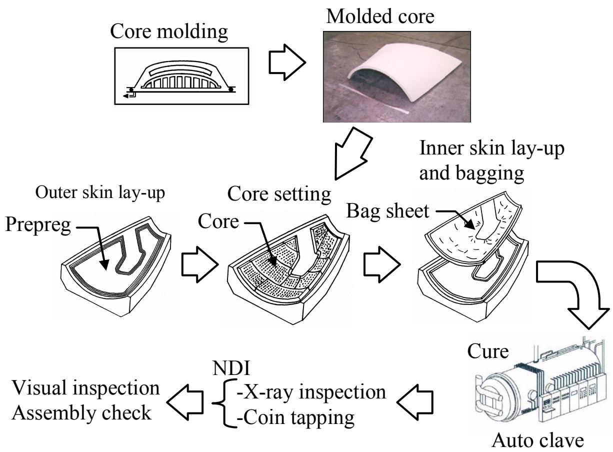

Fig.22 sandwich panel molding process

The major steps are stated below.

1. Core molding process

Ten small cores were molded prior to a sandwich panel fabrication. In this case, solid type core molding jig were used since the multi-support molding jig was still under development.

The amount of the spring-back distortion was very small for each small core.

2. Outer skin lay-up process

CFRP prepregs for the outer skin were stacked. In this case, hand lay-up process was applied because the complicated three-dimensional shape was unsuitable for the application of an automatic lay-up machine.

3. Core setting process

Ten small cores were set on the outer skin prepregs. The foam tape adhesives were installed between each small core.

4. Inner skin lay-up process

On the spliced cores, prepregs of the inner skin were stacked by hand lay-up process.

5. Bagging and cure

The Prepreg and core of the sandwich panel were bagged and cured with an autoclave. The cure cycle was 180 centigrade holding for two hours.

6. Non destructive inspection

As for the non-destructive inspection, X-ray inspection and coin tapping were conducted. Relating to the ultrasonic inspection, the process of the large size sandwich panel is now under study.

4.2.3 Tool try article

The tool try ariticle of 3.5m by 3.0m is shown in Fig.23.

This article was fabricated prior to the test article in order to confirm the validity of the manufacturing process. A visual inspection and non distructive inspection were conducted to evaluate inner quality and manufacturing distortion.

Fig. 23 Tool try article

Through this development, no significant flaw was found. The manufacturing distortion was about 2 or 3 mm comparing with the surface of the molding jig.

This level of distortion is small enough to affect the assembly.

4.3 Full scale test plan

4.3.1 Study of the test article configuration

The test article will be fixed at the rear end. The FEM analysis was conducted to estimate the effect of a constraint. Through this analysis, the effect of the constraint was limited to the local portion near the rear end to the article. Therefore the critical part, such as joint portion and a periphery of windshield cutouts can be evaluated.

The test article configuration is shown in Fig.24.

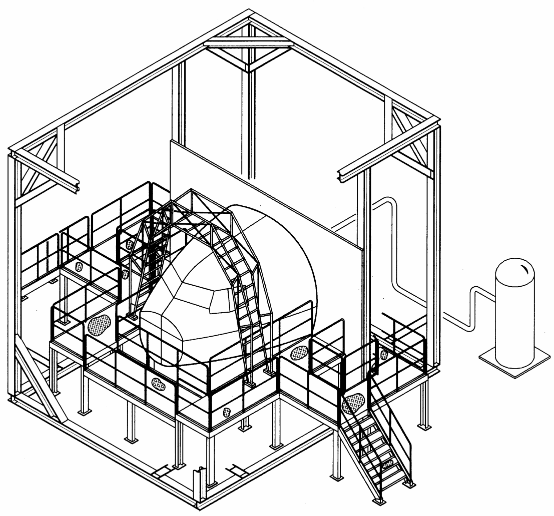

Fig.24 Test article configuration

4.3.2 Test plan

This test will be conducted under the ambient air temperature and pressure to show the compliance with FAR 25.305, 25.307 and 25.843.

(1) Test load

As for the test load, the pressurization load is selected because this load is the critical one for most of the sandwich panel.

The limit load test and ultimate load test whose test load is 1.5 multiplied by the limit load were evaluated.

*Pressurization load : 74.6KPa (limit load)

(2) Item of evaluation

The following items from FAR25.305 will be evaluated.[4]

(a) The structure must be able to support limit loads without detrimental permanent deformation. At any load up to limit loads, the deformation may not interfere with safe operation.

(b) The structure must be able to support ultimate loads without failure for at least 3 seconds.

The strains and deformations of the test article will be measured.

5. Non-destructive inspection

In order to apply this large size CFRP sandwich panel to the actual aircraft, a process of non-destructive inspection is to be established. In case of manufacturing, the process can be set based on the conventional one. For the aircraft maintenance, a new process is necessary. The inspection process suitable for large size parts and airline operation shall be established. A thermography and an ultrasonic inspection are the possible candidates.

6. Structural repair

Structural repair method is also necessary for the actual aircraft application. The process may be set based on the conventional method for a sandwich panel. However, the evaluation by the airline maintenance personnel will be inevitable.

7. Lightning strike protection

As for the lightning strike protection, installations of the cupper expand metal is the candidate. In order to verify the effect of the protection, the verification test is necessary. This test will be conducted in the next step of our development.

8. Conclusion

Through the basic technology establishment phase, the following results are obtained.

1. Nose structural design was finalized. Sandwich panel design method including joint design method was established.
   Through this design, our numerical targets of the 20% weight reduction and 80% part-count reduction can be achieved.
2. Design data of the sandwich panel such as joint and core splice were obtained.
3. Low cost concept for core molding was established with relating manufacturing data.
4. Large size sandwich panel fabrication process was established through the fabrication of tool try article.
5. Full scale structural test plan was prepared.

Based on these results, the next phase of the structural engineering verification phase will be proceeded. Our final goal is to confirm the validity of our design through the full-scale nose structural test.

Reference

[1] Hirose Y, Maekawa S, Kosugi K, Imuta M, Tajima N and Kikukawa H, Technology Development for Innovative Structure of a Transport Aircraft, *Proc Conference Termec2000*, Las Vegas USA, CDROM, Session D6, Vol. 117/3, 2000.

[2] Hirose Y, Tujimoto T, Kosugi K, Imuta M, Kikukawa H, Innovative Nose Section of Fuselage structures Applied by Friction Stir Welding (FSW) of The Aluminum Alloy and Large Thin Casting of Improved Aluminum Alloy, *Proc Conference LiMat2001*, Pusan, Korea, pp489-pp494, 2001

[3] Hirose Y, Konishi M, Kosugi K, Imuta M, Kikukawa H, The Industrial Application of CFRP sandwich panel for Aircraft structure, *Proc Conference ICCE/8*, Tenerife, Canary Island, Spain, pp355-356, 2001

[4] FAR Part 25, revised January 1.1999