МІНІСТЕРСТВО ОСВІТИ І НАУКИ УКРАЇНИ
НАЦІОНАЛЬНИЙ ТЕХНІЧНИЙ УНІВЕРСИТЕТ УКРАЇНИ
“КИЇВСЬКИЙ ПОЛІТЕХНІЧНИЙ ІНСТИТУТ імені ІГОРЯ СІКОРСЬКОГО”

Механіко-машинобудівний інститут
(назва факультету, інституту)

Динаміки і міцності машин та опору матеріалів
(назва кафедри)

"На правах рукопису"
УДК__________

«До захисту допущено»
Завідувач кафедри ДММ та ОМ
__________Пискунов С. О
(підпис) (ініціали, прізвище)

“___” _____________2019р.

МАГІСТЕРСЬКА ДИСЕРТАЦІЯ

зі спеціальності 131 Прикладна механіка
(код та назва спеціальності)

На тему “Міцність композитної конструкції робочого місця бортпровідника пасажирського літака”

Студент групи МП-83мп _Зінькевич Дар’я Андріївна_ __________
(шифр групи) (прізвище, ім’я, по батькові) **(підпис)**

Науковий керівник: к. т. н., доцент Онищенко Є. Є. __________
(вчені ступінь та звання, прізвище, ініціали) **(підпис)**

Рецензент: д. т. н., проф. Данильченко Ю. М. __________
(вчені ступінь та звання, прізвище, ініціали) **(підпис)**

Засвідчую, що у цій магістерській дисертації немає запозичень з праць інших авторів без відповідних посилань.
Студент_______

Київ 2019

Національний технічний університет України

«Київський політехнічний інститут ім. Ігоря Сікорського»

Факультет (інститут) Механіко-машинобудівний інститут .

(повна назва)

Кафедра Динаміки і міцності машин та опору матеріалів .

(повна назва)

Освітньо-кваліфікаційний рівень «магістр»

Напрям підготовки 6.050501 Прикладна механіка .

(код і назва)

Спеціальність 8.05050101 Динаміка і міцність машин .

(код і назва)

ЗАТВЕРДЖУЮ

Завідувач кафедри

__________ С.О. Пискунов

(підпис) (ініціали, прізвище)

«__» ________2019 р.

ЗАВДАННЯ

на магістерську дисертацію студенту

Зінькевич Дар’ї Андріївні .

(прізвище, ім’я, по батькові)

1. Тема дисертації Міцність композитної конструкції робочого місця бортпровідника пасажирського літака

науковий керівник дисертації к.т.н., доцент Онищенко Є. Є. ,

(прізвище, ім’я, по батькові, науковий ступінь, вчене звання)

затверджені наказом по університету від «___» _______ 2019 р. № _______

2. Строк подання студентом дисертації: «___» _______ 2019 р

3. Об’єкт дослідження: робоче місце бортпровідника пасажирського літака

4. Предмет дослідження: аналіз міцності

5. Перелік завдань, які потрібно розробити: змоделювати скінченно-елементну модель у програмному комплексі Patran, проаналізувати отримані результати

6. Орієнтовний перелік ілюстративного матеріалу 41 малюнок

7. Орієнтовний перелік публікацій

8. Консультанти розділів дисертації

Розділ	Прізвище, ініціали та посада консультанта	Підпис, дата	
		завдання видав	завдання прийняв

9. Дата видачі завдання: 02 вересня 2019 року

Календарний план

№ з/п	Назва етапів виконання магістерської дисертації	Строк виконання етапів магістерської дисертації	Примітка
1	Ознайомлення з літературою	02.09.19-16.09.19	Виконано
2	Ознайомлення з програмним середовищем Patran/Nastran	16.09.19-30.09.19	Виконано
3	Розробка скінченно-елементної моделі з метою отримання значень внутрішніх зусиль	30.09.19-21.10.19	Виконано
5	Обробка отриманих результатів	21.10.19-28.10.19	Виконано
6	Розрахунки по аналізу міцності з’єднань панелей робочого місця бортпровідника	28.10.19-18.11.19	
7	Підготовка презентації по стану роботи	18.11.19-25.11.19	Виконано
8	Оформлення роботи	25.11.19-16.12.19	Виконано

Студент ____________ Зінькевич Д. А.

(підпис) (ініціали, прізвище)

Науковий керівник дисертації ____________ Онищенко Є. Є.

(підпис) (ініціали, прізвище)

CONTENTS

Abstract

The master`s degree dissertation for the amount of work is 80 pages, 41 figures, 51 tables, and contains 6 literature.

The object of the work is the personal work station (PWS) of flight attendant for passenger aircraft.

The main goal of this dissertation is the detailed analysis of PWS with safety requirements.

The installation of PWS is analyzed using the finite element analysis software MSC (MSC Patran, MSC Nastran), and also Microsoft Excel.

As a result of this work, it was proved that the personal work station of flight attendant withstands all specified overloads without damage. Safety requirements met.

Реферат

Дана магістерська дисертація за обсягом роботи складає 80 сторінок, 41 ілюстрацію, 51 таблицю та містить 6 літературних джерел.

Об'єктом дослідження є робоче місце бортпровідника пасажирського літака.

Головна ціль даної дисертації – детальний аналіз робочого місця бортпровідника згідно з вимогами безпеки.

Аналіз виконується методом скінченних елементів (МСЕ) за допомогою програмних комплексів MSC Patran, MSC Nastran, а також Microsoft Excel.

В результаті даної роботи було доведено, що робоче місце бортпровідника витримує всі задані перевантаження без пошкоджень, вимоги безпеки виконуються.

1. Formulation of the problem

The flight attendant Personal Work Station is a necessary item for a comfortable flight of passengers. It is cabin structure that has evolved specifically for the purpose of mounting Cabin System Equipment. This is identified as the “Cabin Electronics Compartment” (CEC). The PWS may include the Cabin In-Flight Entertainment equipment, terminals, etc (Figure 1.1).

Figure 1.1. Flight attendant Personal Work Station. Photo.

In twin-aisle models, the CEC may stretch from floor to ceiling in the cabin, or may be installed in the crown area above the cabin ceiling, or may be as large as will fit under the stairs to the upper deck. On single-aisle models, the CEC may be small enough to fit in an overhead stow bin.

Cabin System equipment installed in main deck CECs generally includes those units which require regular attention by the cabin crew. Cabin System equipment installed in overhead CECs includes only those units which do not require attention by the cabin crew.

Stand-alone PWS, extending from floor to ceiling, will be covered in this dissertation. Generally, typical stand-alone PWS structure is attached to the airplane floor via fittings and to the overhead aircraft structure by means of a tie rod. But due to the interior’s features of the specific aircraft, attachment at the top is impossible. The customer demanded to calculate the PWS, provided only attachment to the floor.

Due to the lack of a top attachment, overloads will produce significant values of the moments compared to moments in a typical PWS structure, that will cause large deformations and displacements. The task is to analyze the new type of PWS (without top attachment) for compliance with safety requirements and to prove that all margins of safety are positive.

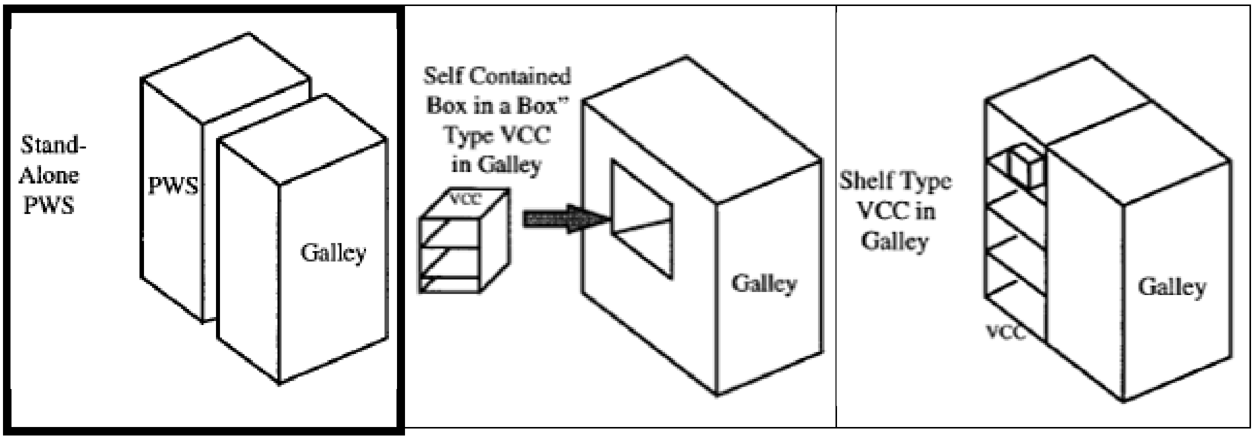


Figure 1.2. Types of PWS.

2. Introduction

The analysis of the flight attendant's personal work station presented in this dissertation is performed using the finite element method (FEM).

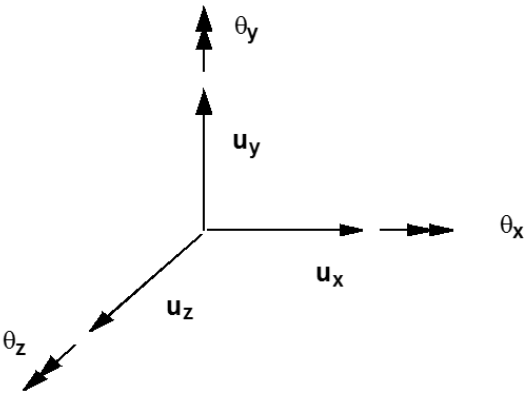
2.1. Finite Element Method history

The finite element method originated from the need for new ways to solve complex elasticity and structural analysis problems in civil and aeronautical engineering in the 1930s. The founders of the ideas underlying the FEM are considered Alexander Hrennikoff and Richard Courant. Their researches were published in the 1940s. For the first time, the effectiveness of FEM was demonstrated in 1944 by Ioannis Argyris, who implemented a computer-based method. Further development of the finite element method is also associated with the solution of space research problems in the 1950s. In the USSR, the introduction of the practical application of the method is usually connected with name of Leonard Oganesyan. The finite element method obtained its real impetus in the 1960s and 1970s. Its study involved scientists from the University of Stuttgart, the University of California at Berkeley, the Swansea University, the Cornell University, etc.

At the moment, FEM is one of the most effective modern methods for the numerical solution of engineering, physical, and mathematical problems using computers. The capabilities of the method are constantly expanding with the development of computing tools, and the type of tasks to be solved is also expanding. At present, a large number of implementations of the finite element method are proposed for modeling diffusion, heat conduction, hydrodynamics, mechanics, electrodynamics.

2.2. Finite Element Method description

The finite element method is a numerical approximation method for solving problems of engineering and mathematical physics. It is a method of investigating the behavior of complex structures by breaking them down into smaller, simpler pieces with the same physical and mechanical properties as the considered structure. The main method's idea is the discretization of a continuous area by a mesh into a set of discrete subdomains, usually called elements. It is assumed that these elements are connected to each other at the nodes. Each node is capable of moving in six independent directions: three translations and three rotations. These are called the degrees of freedom (DOF) at a node.



Three translations (u_x, u_y, u_z).

Three rotations (θ_x, θ_y, θ_z).

{u} – displacement vector

$\{u\} = \{u_x\ u_y\ u_z\ \theta_x\ \theta_y\ \theta_z\}$

Figure 2.1. Degrees of freedom (DOF) at node.

The assembly of elements and nodes is called a finite element model. Finite elements have shapes which are relatively easy to formulate and analyze. The three basic types of finite elements are beams, plates, and solids (Figure 2.2).

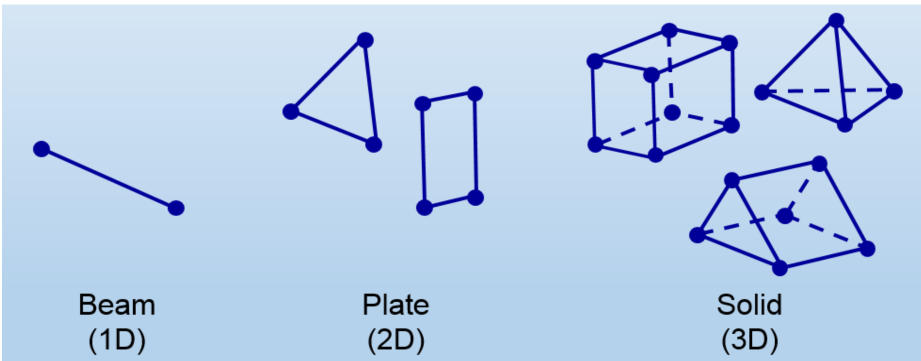


Figure 2.2. Types of finite elements.

A stress-strain state for each finite element is studied by known structural analysis methods at the points of connection. There are two main methods – the displacement method and the force method. Forces or displacements are accepted as the main unknown finite element method. To find them, algebraic equations are compiled. These simple equations are then assembled into a larger system of equations that models the entire problem, and then the system is solved.

The main equation for the displacement method is:

$$\{F\} = [K]\{u\}$$

Where:

$\{F\}$ – forces acting on the structure;

$\{u\}$ – displacements resulting from {F};

$[K]$ – stiffness matrix [k_{ij}], where each k_{ij} term is the force of a constraint at coordinate *i* due to a unit displacement at *j* with all other displacements set equal to zero.

However, usually finite element analysis (FEA) is difficult for hand calculation and involves solution of engineering problems using computers. Engineering structures that have complex geometry and loads, are either very difficult to analyze or have no theoretical solution. However, in FEA, a structure of this type can be easily analyzed. Commercial FEA programs, written so that a user can solve a complex engineering problems without knowing the governing equations or the mathematics; the user is required only to know the geometry of the structure and its boundary conditions. FEA software provides a complete solution including deflections, stresses, reactions, etc.

FEA solution of engineering problems, such as finding deflections and stresses in a structure, requires three steps:

- pre-process or modeling the structure;
- analysis;
- post processing.

Step1: Pre-process or modeling the structure

Using a CAD program that either comes with the FEA software or provided by another software vendor, the structure is modeled. The final FEA model consists of several elements that collectively represent the entire structure. The elements not only represent segments of the structure, they also simulate it is mechanical behavior and properties. Regions where geometry is complex (curves, notches, holes, etc.) require increased number of elements to accurately represent the shape; where as, the regions with simple geometry can be represented by coarser mesh (or fewer elements). The elements are joined at the nodes, or common points.

In the pre-processor phase, along with the geometry of the structure, the constraints, loads and mechanical properties of the structure are defined. Thus, in pre-processing, the entire structure is completely defined by the geometric model. The structure represented by nodes and elements is called “mesh”.

Step 2: Analysis

In this step, the geometry, constraints, mechanical properties and loads are applied to generate matrix equations for each element, which are then assembled to generate a global matrix equation of the structure. The form of the individual equations, as well as the structural equation is always $\{F\} = [K]\{u\}$.

The equation is then solved for deflections. Using the deflection values, strain, stress, and reactions are calculated. All the results are stored and can be used to create graphic plots and charts in the post analysis.

Step 3: Post processing

This is the last step in a finite element analysis. Results obtained in step 2 are usually in the form of raw data and difficult to interpret. In post analysis, a CAD program is utilized to manipulate the data for generating deflected shape of the structure, creating stress plots, animation, etc. A graphical representation of the results is very useful in understanding behavior of the structure.

3. PWS location and structure

The PWS is located in front of the passenger compartment, aft of passenger entry door on the left side, as shown in the Figure 3.1.

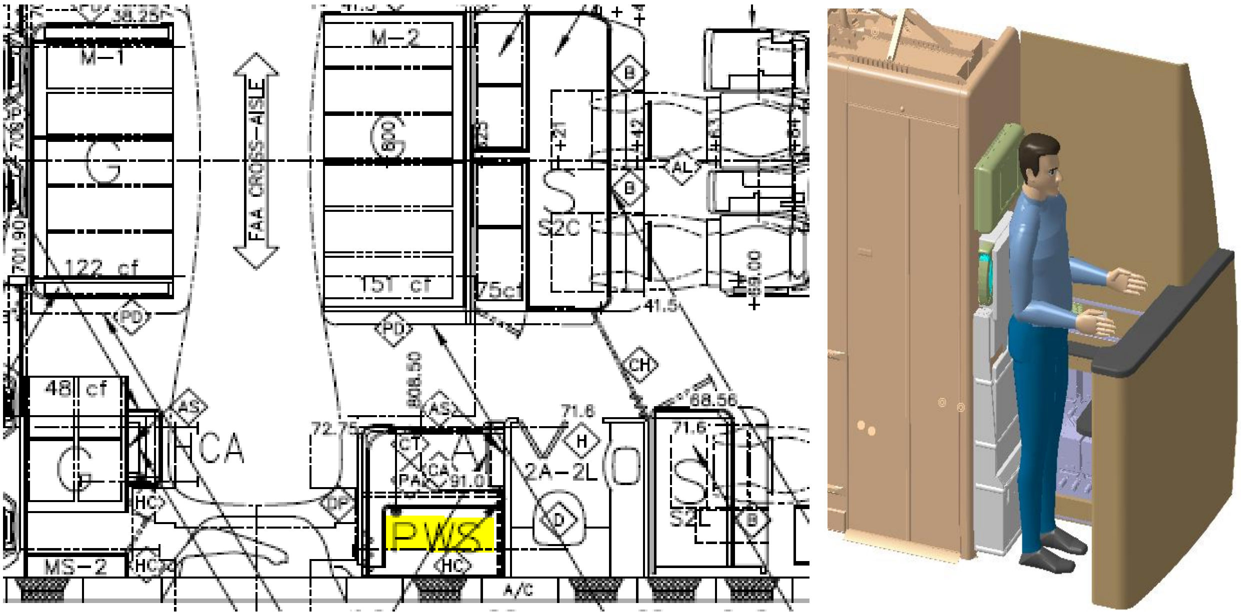


Figure 3.1. Layout of passenger accommodation (LOPA).

The PWS is attached to the airplane floor via 4 fittings. The two floor fittings (points 4 and 5) are slotted in the “Y” of sideways direction (VF type) and are attached to the aft and forward vertical panels of the PWS. The two other floor fittings (points 1 - 3) are circular (VSF type) and are attached to the floor panel and forward vertical panel. There is not an Attendant Seat at the PWS. The PWS structure is shown in Figure 3.2.

The PWS weighs is 384.29 lbs. The panels are honeycomb sandwich panels with fiberglass factsheets with a thickness of either 1 inch, 0.50 inch, 0.52 inch or 0.55 inch.

The list of Panels, Metal parts, Equipment and other elements with appropriate weights is presented in Table 3.1.

Item	Title	Thickness (in)	Weight (lb)
Panel Assemblies	PANEL ASSY, FLOOR	1.00	10.17
	PANEL ASSY, Lower Aft	1.00	2.89
	PANEL ASSY, Lower OUTBD	0.50	4.75
	PANEL ASSY, Lower INBD	1.00	2.89
	PANEL ASSY, Work Surface	1.00	4.94
	PANEL ASSY, Lower FWD	1.00	10.03
	PANEL ASSY, Upper bottom	1.00	2.38
	PANEL ASSY, Upper Aft	0.52	2.15
	PANEL ASSY, Upper OUTBD	0.50	6.86
	PANEL ASSY, Upper INBD	0.52	1.39
	PANEL ASSY, Upper FWD	0.52	1.87
	PANEL ASSY, Lower Bottom Plenum	0.50	1.73
	PANEL ASSY, Lower Top Plenum	0.50	3.27
	PANEL ASSY, Door Maint Access	0.50	0.25
	PANEL ASSY, Door Hatch	0.50	1.93
	PANEL ASSY, Door Printer & Switches	0.50	0.83
	PANEL ASSY, Shelf Misc Stowage base	0.50	1.50
	PANEL ASSY, Misc stowage	0.50	2.66
	PANEL ASSY, Shelf Misc Stowage	0.50	0.93
	PANEL ASSY, Upper Instl access	0.55	1.13
	PANEL ASSY, Upper maint. Access	0.55	3.94
	PANEL ASSY, Door Misc Stowage	0.50	0.62
	PANEL ASSY, Upper Plenum	0.52	1.70
	BFE DÉCOR PANEL, UPPER	0.42	9.42
	BFE DÉCOR PANEL, LOWER	0.42	8.91
Metal Parts	INBD AFT Seat Track bracket		0.40
	CSCP Shroud		0.84
	Counter Top		12.21
	Switch Closeout		3.58
	GND TEST ENABLE BOX		0.14
	OUTBD FWD Seat Track		0.43
	OUTBD AFT Seat Track		0.43
	INBD FWD Seat Track		0.48
	Consul		2.79
	Pallet access .1		1.61
	Pallet access .2		1.61
	Pallet access .3		1.61
	Switch Panel		0.60
	OUTBD Access		0.13
	CSCP BRCKT .1		0.25
	CSCP BRCKT .2		0.25
	BRKT FWD Upper Acess(2)		0.08

	Threshold SS	0.57
	Light Panel	0.60
	BFE UPPER RUBSTRIP	0.17
	BFE LOWER RUBSTRIP	0.17
	DÉCOR ATTACHMENT HARDWARE	1.33
Miscellaneous Items	Foam	60.18
	Dog Bones	
	Inserts and potting	
	Rub Strips, Kick Strips, Trim	
	Seat Track Fittings	
	Fasteners	
	Spud Assy and Fittings	
	Grilles	
	Overhead Support	
	Hinges and Latches	
	Light, Switches, Wiring	
	5% Margin	
IFE Equipment	CREW TERMINAL(CT)	4.90
	CT Shroud	1.25
	SPM	3.85
	SPM RACK	0.40
	PRINTER	11.30
	PRINTER RACK	1.90
	POWER OUTLET 1	0.13
	POWER OUTLET 2	0.13
	Pallet 1 UNLOADED 35 lbs	81.40
	Pallet 2 UNLOADED 17 lbs	49.20
	IFE Handset	0.49
	IFE Handset Cradle	0.75
Cabin Systems Equipment	ATTENDANT HANDSET CRADLE	1.25
	ATTENDANT HANDSET	1.25
	CSCP MONITOR	18.70
Stowage Shelves	Stowage Shelf 1	10.00
	Stowage Shelf 2	10.00
Wiring	Wiring (5% of Equipment Weight)	7.79
Total / Sum	**384.29**	

Table 3.1. List of Panels, Metal parts and Equipment of PWS.

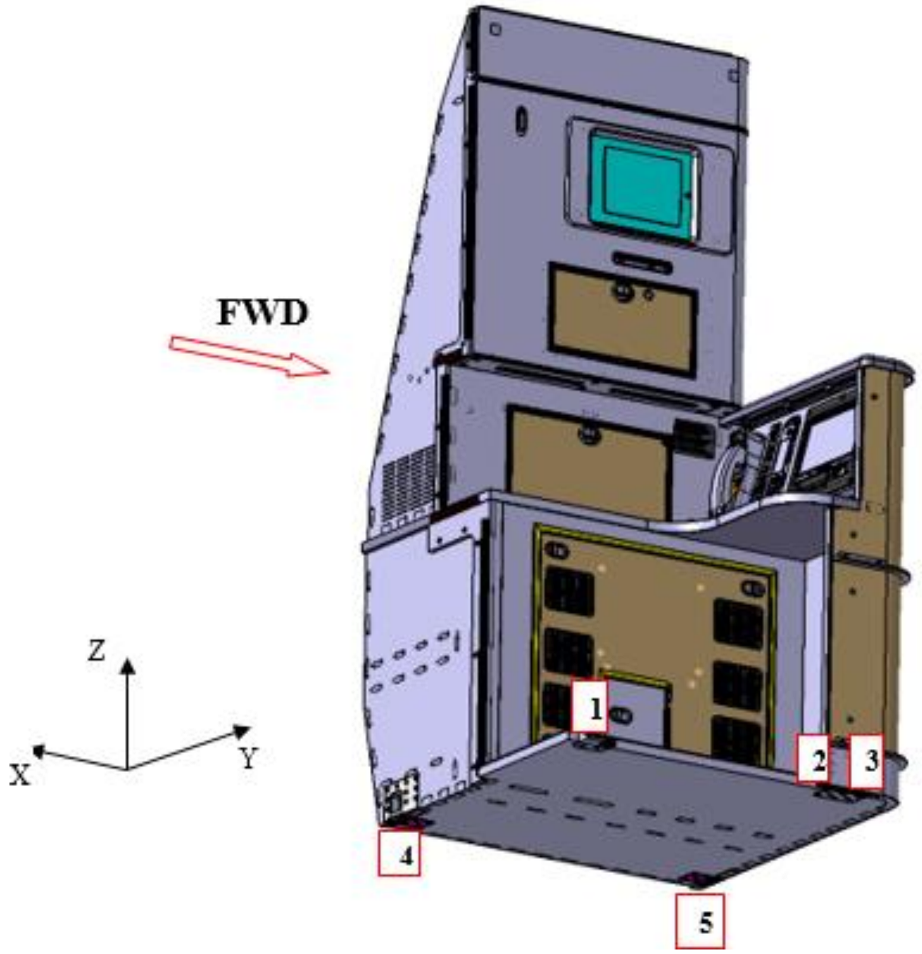


Figure 3.2. PWS structure.

4. Finite element model

4.1. Types of elements used in the model

The installation of PWS is analyzed using the finite element analysis software MSC (combination of MSC PATRAN and MSC NASTRAN). For an accurate analysis by FEM, selection of the proper elements is very important. The selected elements must represent the engineering structure as close to the original structure as possible. In this analysis, shell elements (QUAD4) are used to model the Parts of PWS. MPC elements (RBE3) are used to simulate the Equipment. BUSH elements, BAR elements, MPC (RBE2) are used to simulate the joints between the Parts, such as Dog-Bone connection, Tab & Slot connection, etc.

The PWS has been analyzed as a stand-alone model with no load share from other models. A Finite Element Model without doors is presented in the Figure 4.1.

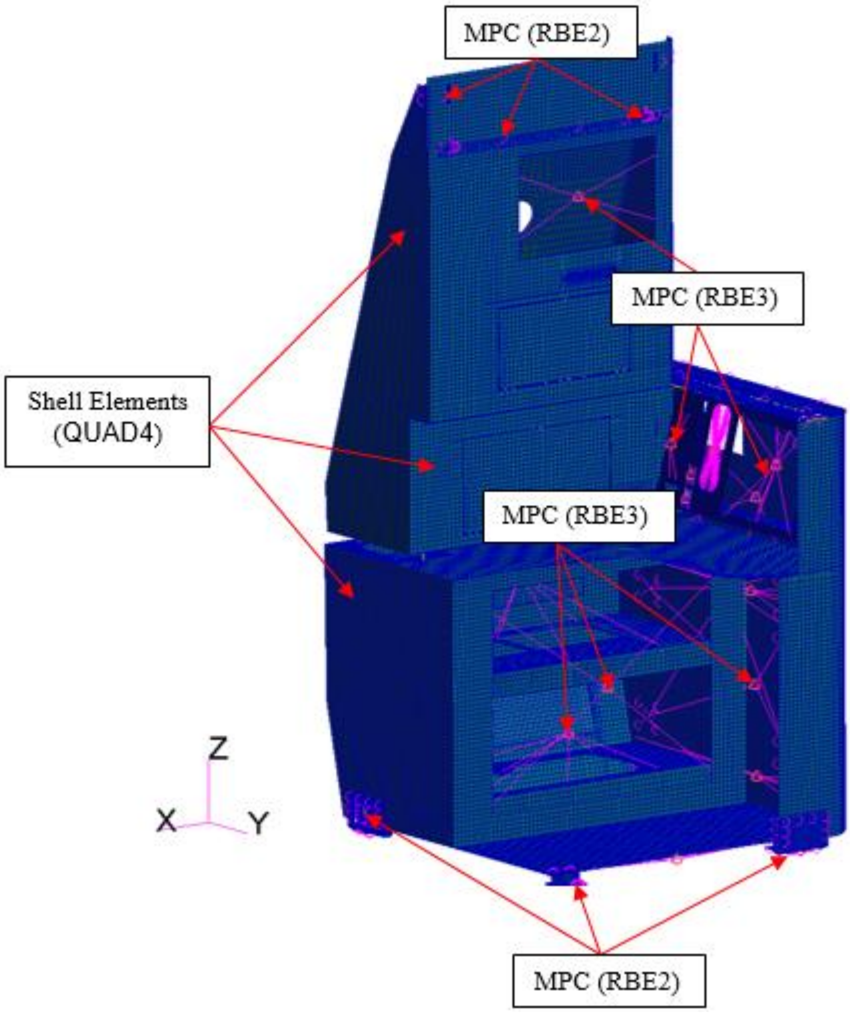


Figure 4.1. PWS Finite Element Model.

The PWS Panels are jointed to each other using Dog-Bones and Tab & Slot connections. Dog-Bones are scalloped single or double shear-tie fittings really shaped like a bone (hence the name). The Dog-Bone is presented in the Figure 4.2.

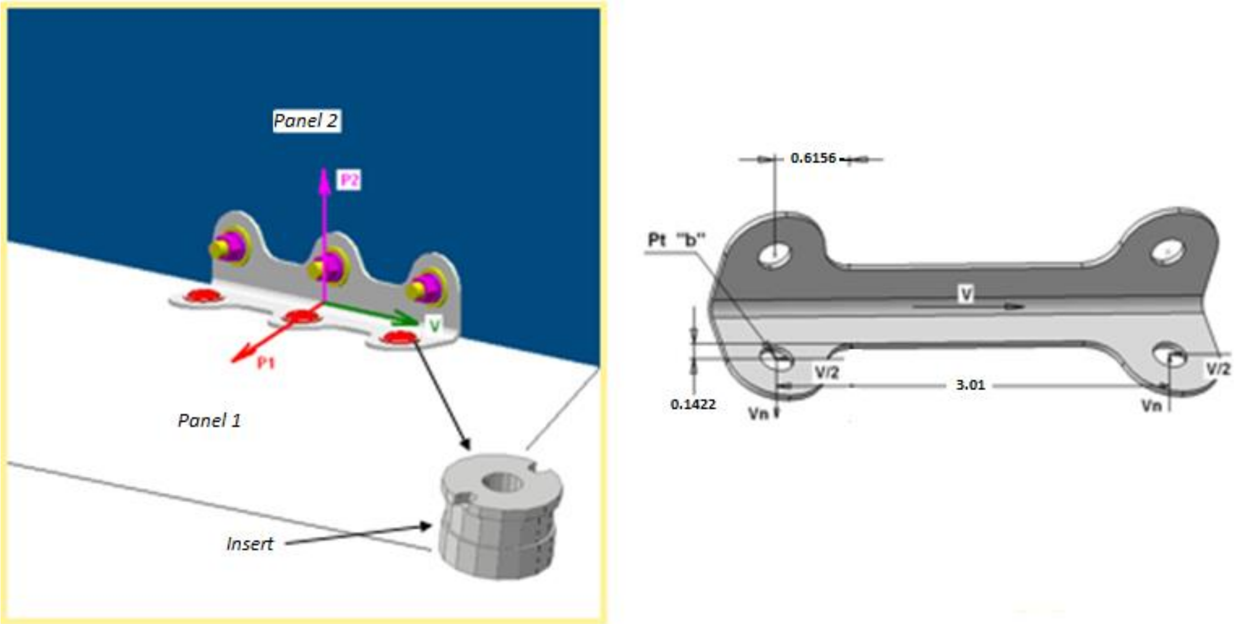


Figure 4.2. Dog-Bone.

To create a Tab & Slot connection, a groove is generated and the panel is bent along the groove. The panel includes a first skin, second skin, and core. The core is sandwiched between the first skin and the second skin. The groove passes through the first skin and at least a portion of the core. The groove includes a set of tabs and a corresponding set of slots. The set of tabs intermesh with the set of slots in response to bending the panel along the groove. The Tab & Slot connection and its modeling in MSC Patran is presented in the Figure 4.3.

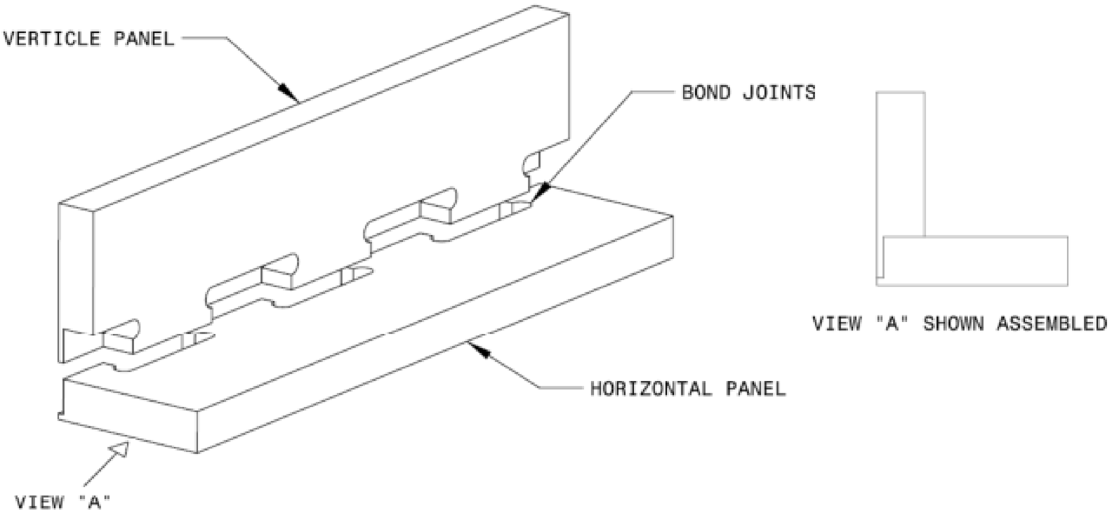


Figure 4.3. Tab & Slot connection.

QUAD elements are two-dimensional quadrilateral elements, commonly referred to as plate and shell elements, are used to represent areas in model where one of the dimensions is small in comparison to the other two, the thickness is substantially less than dimensions a or b (Figure 4.4). These elements capable of carrying inplane force, bending forces, and transverse shear force. This family of elements are the most commonly used 2-D elements in the MSC Nastran element library. QUAD elements are preferred over the triangular elements (TRIA) for accuracy reasons. The latter are mainly used for mesh transitions or for modeling portions of a structure when quadrilateral elements are impractical.

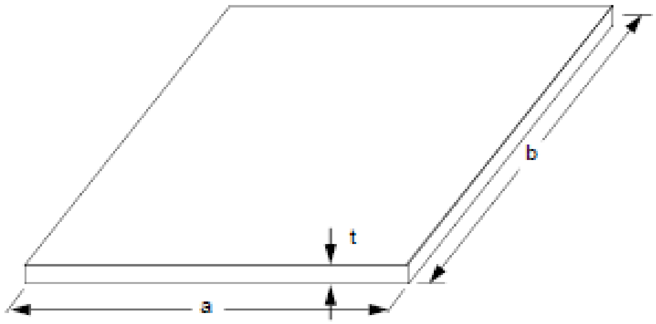


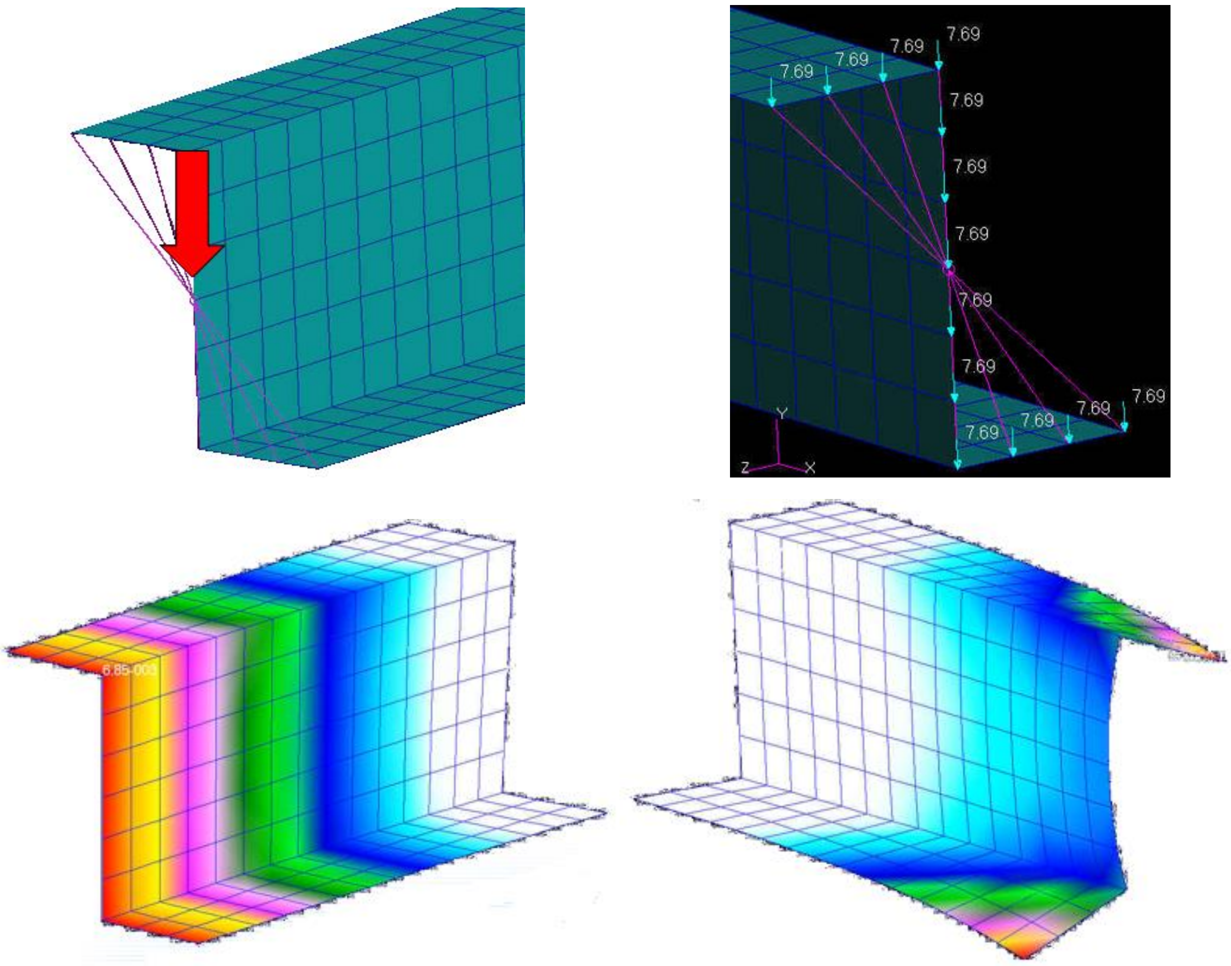
Figure 4.4. QUAD element.

MSC Nastran offers various ways of modeling structural connections and fasteners. Bolts, screws, and so on can be represented, depending on the modeling goals, either with flexible springs or bars (BUSH, BAR), rigid elements (RBE2, RBE3), or multipoint constraints (MPC). Connections can be established with ease between points, elements, patches, seam lines, dissimilar meshes, or any of their combinations. The connector elements are general in purpose, easy to generate and always satisfies the condition of rigid body invariance.

MPC (RBE2, RBE3) – Rigid Body Element is often used to connect one node to several nodes, when we must distribute one load to several nodes. In another case RBE2 allows modelling absolutely rigid fasteners in assembles, when we can neglect fasteners ductile. RBE2 elements enforce beam theory (plane sections remain planar). RBE3 elements aren't absolutely rigid and allow warping.

The forces / moments applied to one node are distributed among the other in same manner as classical bolt pattern analysis. Mass is distributed among the nodes according to their weighting factors.

The example of the deformation of a beam loaded with a transverse force (defined using RBE2 or RBE3) is presented below. When using RBE3 element the uniform load distribution results in too much transverse load in flanges causing them to droop. The quadratic distribution of load in the web can be used to more realistic display of load distribution.



RBE2 **RBE3**

Figure 4.5. RBE2 / RBE3 comparison.

BUSH elements is a generalized spring-damper elements. They relate to structural scalar elements. BUSH element connects two non coincident grid points, or two coincident grid points or one grid point. The BUSH avoids the internal constraint problem.

BAR elements are straight one-dimensional elements that connect two grid points. The one-dimensional elements are used to represent structural members that have stiffness along a line or curve between two grid points. Typical applications include beam type structures, stiffeners, tie-down members, supports, mesh transitions, and many others.

BAR elements are capable to taking axial, bending and shear loads. For these elements, it is necessary to create geometric and material properties. The capabilities and limitations of BAR elements are presented below.

• Extensional stiffness along the neutral axis and torsional stiffness about the neutral axis may be defined.

• Bending and transverse shear stiffness can be defined in the two perpendicular directions to BAR element’s axial direction.

• The properties must be constant along the length of BAR element.

• The shear center and the neutral axis must coincide.

• The ends of BAR element may be offset from the grid points.

• The effect of out-of-plane cross-sectional warping is neglected. This limitation is not present in BEAM element.

• The stress may be computed at up to four locations on the cross section at each end.

• Transverse shear stiffness along the length of BAR can be included.

Calculation with BAR elements requires more computing power then RBE2 elements, because every BAR element has 6 DOF and every RBE2 element has 3 DOF, but BAR element gives more accurate result.

4.2. Construction of finite element model

The finite element model in Patran is created based on the CATIA model. Firstly, a midplane outline of each panel is created except for the ceiling and floor panel. For the ceiling panel, use the top surface. For the floor panel, use the bottom surface. To make a midplane outline, the "Extract" button must be used to get a surface then translate or offset the surface half the thickness of the panel.

Then a wireframe of dog-bones is created by creating the mid-point of single, double or triple tab dog-bones to represent the dog-bones jointed of the panels together. These points are created as follows:

- for single tab, a point at the center of one of the holes must be created, and then the point is projected to the connection of the two midplane surfaces;
- for double tab, a point must be created at each of the 2 holes, and then a curve should be created connecting these two holes; after that, a point on middle point is created and projected to the connection of the two midplane surfaces;
- for triple tab, a point must be created at the center of the middle tab, and then it is projected to the connection of the two midplane surfaces.

Also wireframe of insert fasteners and equipment attached points must be created by creating points at these locations, and then they are projected to the appropriate midplane surfaces.

The next step is import the created points and surfaces from CATIA to Patran. For this it is necessary to save the CATIA file as an Iges file (with .igs extension). Once the Iges file is saved, it is inserted into Patran. For importing a solid, should be used STP (step file) option instead of IGES.

For easy meshing and analysis, a group for each specific panel was created. Each group was assigned a unique name for easy and quickly find the required panel. Each group includes the geometry, dogbones and insert points associated with that panel.

Then a mesh seed of 0.5 in – 1.0 in was created on all surfaces. After that, each surface was divided into a mesh using the IsoMesh function. A global length value was taken within 0.5 in - 1.0 in. To force a mesh to go through the specific points, the associate function was used to connect relevant points to the surfaces.

To verify of mesh quality, the following verifications were carried out:

- verification of boundaries of each panel: the whole panel mustn't be divided in several parts;
- verification of duplicates: the same element mustn't be duplicated;
- determination of Jacobian Ratio: this value has to be less than three;
- verification of the normals direction: all elements in the same panel must have the same normals direction.

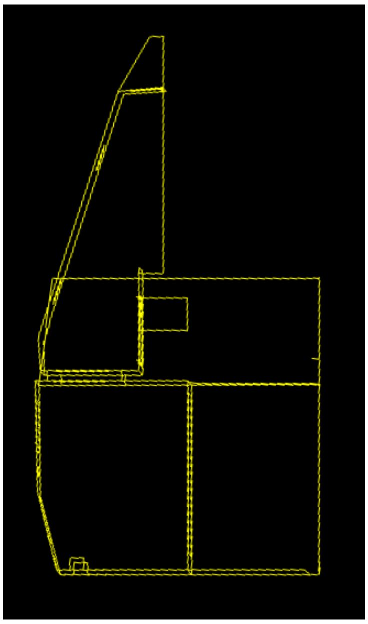


Figure 4.6. Boundaries verification.

Next, a Tab & Slot connection was modeled for panels that have common tabs and slots. For this, BAR elements were used that connected two nodes on two panels at the same location. New properties set was created for the BAR elements by using 1D-Bush. The stiffness was set equal to Kx = Ky = Kz = 1E+8 lb/in and Krot = 24,000 lb/in.

For Dog-Bones connections, BUSH elements similar to the Tab & Slot connection were used. The stiffness for a set of dog bone properties is Kx = Ky = Kz = 10,000 lb/in.

For fastener connections, BUSH elements similar to the Tab & Slot connection were also used. The stiffness equal to the same values as for Dog-Bones connection (Kx = Ky = Kz = 10,000 lb/in).

Next, Door to Panel connection was modeled. This connection includes Door Hinge, Dead bolt latch, and Slam latch:

- all the nodes common to the hinge side of the door and the panel were joined using CBUSH elements with stiffness Kx = Ky = Kz = 5,000 lb/in;
- Dead bolt latches can transfer load in two directions. Latches were modeled using CBUSH elements with stiffness K = 1E+8 lb/in for each direction; the

element was created at the latch's pin/bolt location to connect the door to the surrounding structure;

- Slam latched can transfer load only in one direction; latches were modeled using CBUSH elements with stiffness K = 1E+8 lb/in for this direction.

Using RBE3 element, MPC connection was made for all equipment attachment locations. The center of gravity of an equipment was used as the dependent node and the attachment points were used as the independent nodes. For the dependent nodes, all 6 degrees of freedom are selected, and only the 3 translation DOF are selected for the independent nodes.

Next step is select of boundary conditions. Fixed constraint (6DOF grounded) was used for aircraft interface points. Four fixed constraints were added in places where the fittings are attached to the floor.

After that, loads and inertial conditions were applied. As a loads, the weight of an equipment was used that was applied at the center of gravity of an equipment. If stowage is not attached to any part of the PWS, then it should be modeled as distributed load applied to the panel. Critical panel was selected for each load case. 2D elements were chosen for Target Element Type and then the weight of the stowage was inputted for Surface Load. Inertial conditions were applied to the entire model. There are no nodes applied for the inertia loads. For inertial loads, now factor of 1 was used since later the load factor can be applied to the results.

Further, materials are required for all elements. For metal items, Isotropic materials were used. But since panels of the PWS is honeycomb, Composite materials were used for them. The example of specifying a Composite material shown in the Figure 4.7.

Stacking Sequence Definition

	Material Name	Thickness	Orientation	Global Ply ID
1	Fiberglass	1.100000E-2	0.000000E+0	
2	Fiberglass	1.100000E-2	0.000000E+0	
3	Core_.5_thk	5.000000E-1	0.000000E+0	
4	Fiberglass	1.100000E-2	0.000000E+0	
5	Fiberglass	1.100000E-2	0.000000E+0	

Figure 4.7. Example of specifying composite material in Patran.

Then, a set of properties was specified for each panel and equipment. Panels require material, thickness and nonstructural mass.

Nonstructural mass = (Wp + Wmisc)/Ap, where:

Wp – weight of the panel;

Wmisc – weight of the misc. items (usually Wmisc = Wp*1.05);

Ap – area of the panel.

If the panel formed of an angle of horizontal and vertical, it is necessary to create two separate sets of properties for Material Orientation. An equipment was modeled by Point Elements (0D-Mass). These elements required only a mass of the equipment.

And the final step before carrying out the analysis in Nastran was creation of Load Cases. Three types of Load Cases were added: 1G Forward, 1G Down, 1G Side. Factor of 1 was used and the applicable load factors were applied directly to the final results. Each Load Case includes the applicable boundary conditions, loading conditions and inertial load. The load factors applied to the results depended on the maximum values of overloads experienced by the aircraft during takeoff, flight and landing. These values are presented in FAR (Federal Aviation Regulations). Company Boeing has developed its own overloads values based on FAR value. Both are shown in the Table 4.1.

Load Attitude	FAR (14CFR) requirements	Boeing requirements
Forward	9.0 G	9.0 G
Down	6.0 G	6.5 G
Side	3.0 G	3.0 G
Up	3.0 G	3.5 G
Aft	1.5 G	1.5 G
Down + Fwd	——	6.2G Down + 0.5G Fwd

Table 4.1. Values of overloads, based on FAR / Boeing requirements.

Once all the necessary is set we can start analysis in Nastran. To do this a .bdf file was recorded by Patran. Linear Static was selected as Solution Type. The following were chosen in Output Requests:

- Displacement(SORT1,REAL)=All FEM
- SPCFORCES(SORT1,REAL)=All FEM
- OLOAD(SORT1,REAL)= All FEM
- GPFORCE=All FEM

The .bdf file was run by Nastran. As a result, we got the following file types: .f04, .f06, XDB, etc. In case the model does not run successfully, the .f06 file must be checked for any errors. The errors should be fixed and the .bdf file run one more time. This procedure must be done until no errors are found and the model runs successfully.

To check results, first the results file (XDB) was attached. Next, on the Result tab, we can select the appropriate result to view. Examples of displacements for different overloads are shown in the Figures 4.8 – 4.10.

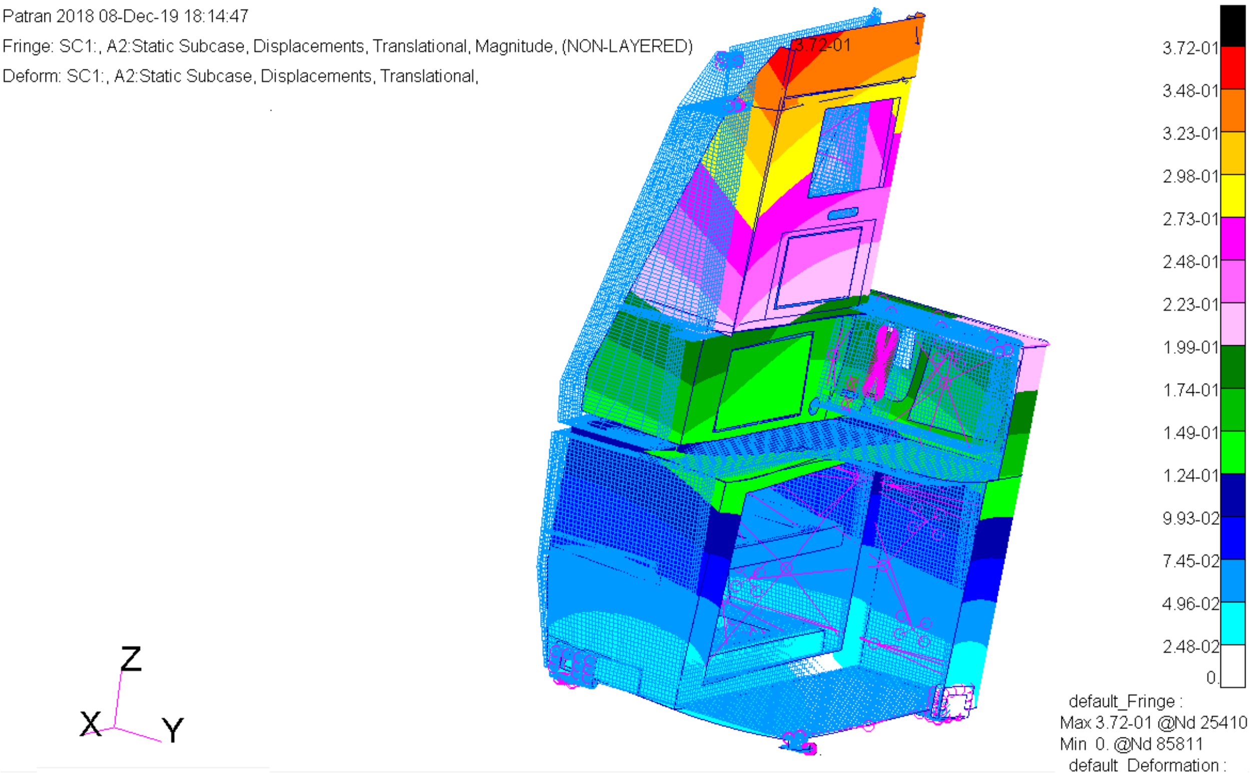


Figure 4.8. PWS displacements for overload ***1G Forward.***

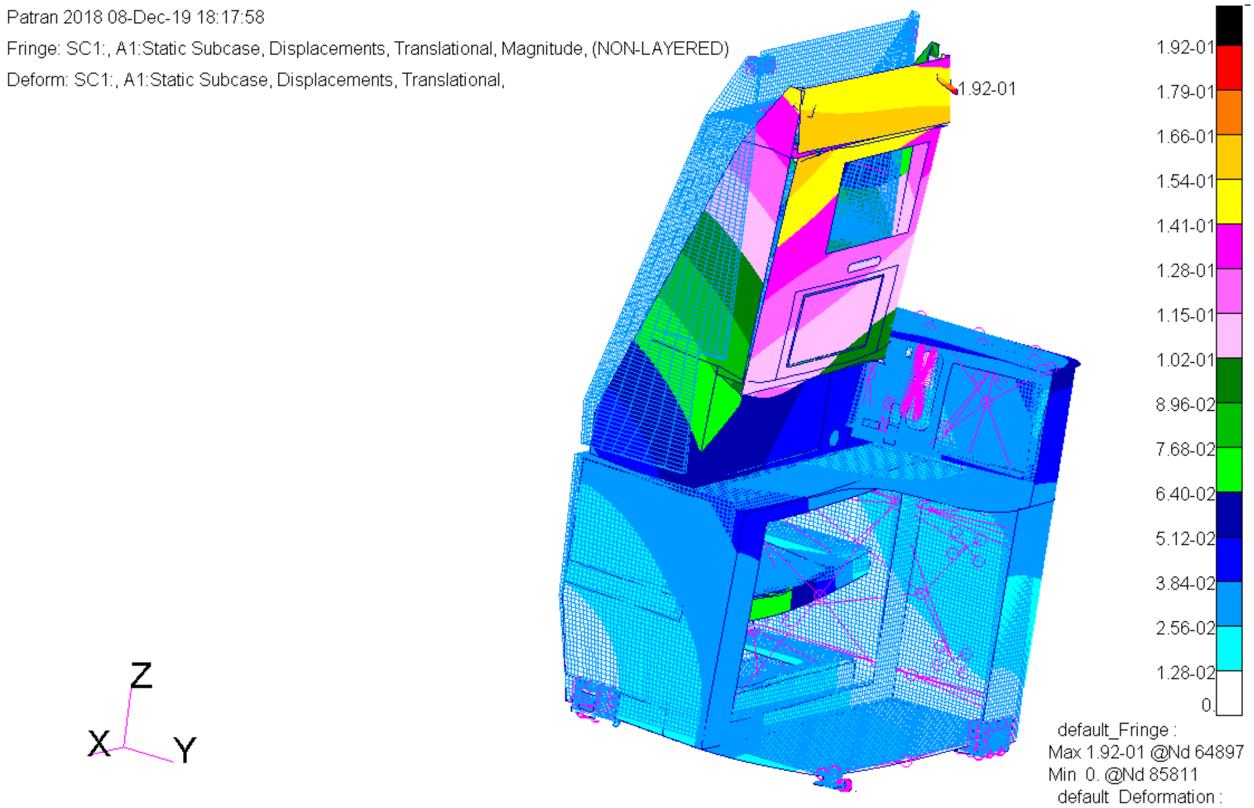


Figure 4.9. PWS displacements for overload ***1G Down.***

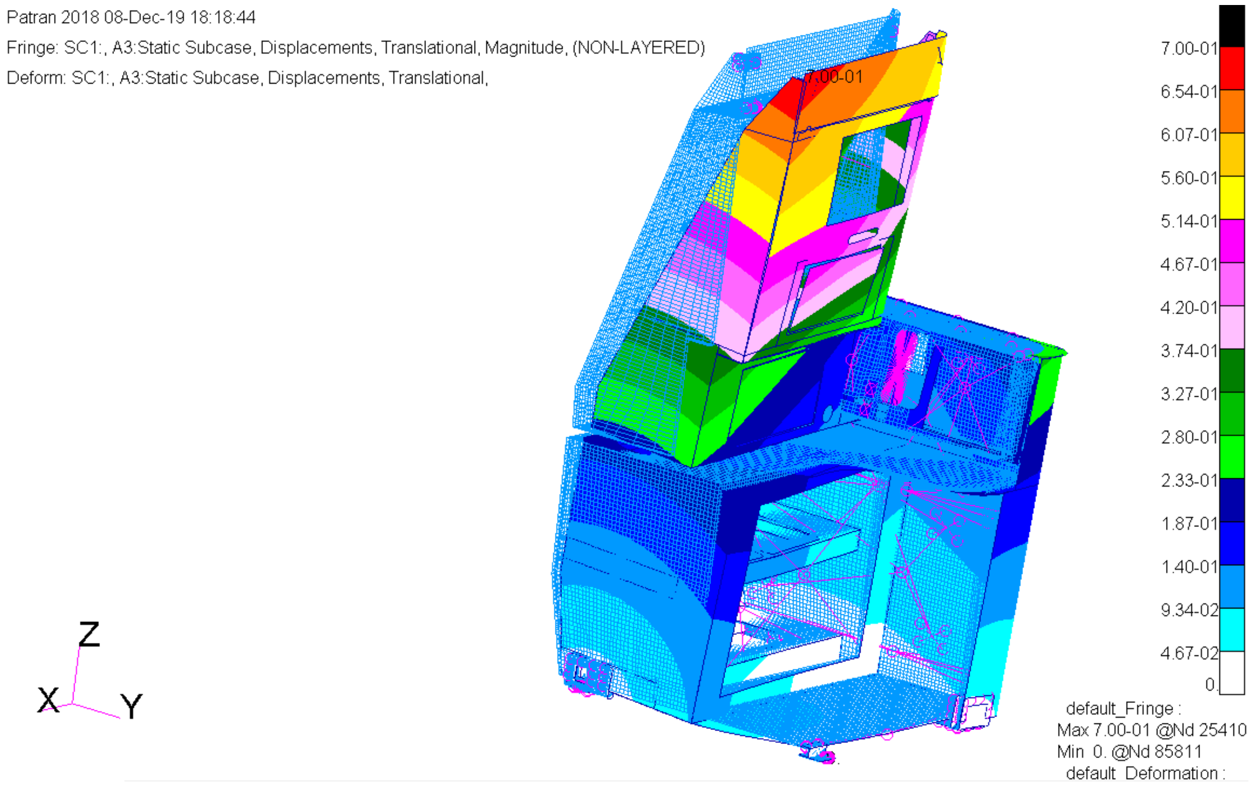


Figure 4.10. PWS displacements for overload ***1G Side.***

A Finite Element Model was built to obtain internal loads for subsequent analysis of panels assembly. Internal loads were extracted from FEM. To get loads for Dog-Bones, Tab & Slot connections, Inserts, the Grid Point Force Balance section from the .f06 file was used. Further analysis was performed using Microsoft Excel.

5. Analysis

Each panel of PWS was analyzed separately to determine Margins of Safety for each fastener. This dissertation presents the results of analyzes for the most loaded panels.

A margin of safety is a measure of the remaining load carrying capacity of a structure existing under an applied load condition. Margins of safety can be determined with respect to virtually any type of criteria.

The form of the standard margin of safety equation is:

$$M.S. = \frac{P_{all}}{P_{app}} - 1$$

where:

P_{all} = allowable load; P_{app} = applied load.

A margin of safety typically serves two functions. First, the algebraic sign of the margin of safety indicates whether or not the applied loads are safe with respect to the allowable loads. Second, the magnitude of the margin of safety indicates the amount the applied loads can be increased without exceeding the stipulated yield or ultimate allowable load. It is important to remember that because of the often non-linear relationship between the applied loads and the resulting stresses, a margin of safety calculated with respect to the loads, in general, will not be the same as a margin calculated with respect to the stresses.

Specifically, margins of safety calculated from stresses will correspond to those calculated from loads only in those cases where a linear relationship exists between the applied loads and resulting stresses up to failure, or at least up to the level of the allowable load. Correspondingly, redundant systems that display redistributions of stiffnesses, systems that operate in the post buckled range or possess other nonlinear behaviors will typically display margins of safety based on loads that are different from those calculated from the stresses. Only in cases where a linear relationship exists between loads and stresses can a margin calculated with respect to stresses provide an accurate margin of safety value. In general, a margin calculated with respect to stresses is not equal to the margin of safety.

But sometimes, when calculating stresses in the members of a structure, it is desirable to compare the stresses due to the applied loads to the member allowable stresses. This is usually done by calculating the stress margin (S.M.) for the member:

$$S.M. = \frac{\text{Allowable stress}}{\text{Applied stress}} - 1$$

A positive S.M. indicates that the member is sufficiently strong to carry the load. That is, the member is acceptable, but the magnitude of the stress margin does not indicate how much the structural loads can be increased prior to failure.

Calculation of stress margins for each member of a complex structure may be useful in determining the ultimate load capacity of the structure which can be used to obtain the margin of safety for the structure (note that a structure fails when its weakest member fails). To obtain the ultimate load capacity of the structure, hence, the load may be increased step by step and the stress margin for each member may be calculated at each load step. As soon as the stress margin for a member becomes equal to zero, the applied load is equal to the ultimate load capacity of the structure, and the weakest member is at the onset of failure. Knowing the original applied load and the ultimate load capacity of the structure, the margin of safety of the structure can easily be obtained from the standard margin of safety equation.

In this analysis, each structural member in question is subjected to combined loads. In this case, the failure can be determined by using the interaction method. The interaction method uses curves and/or equations that represent the loading condition at yield or at failure of a structural member subjected to two or more simultaneously applied loads. The interaction curves and/or associated equations are determined by test or theory for a structural member under combined loading.

The applied load and the allowable load are used to calculate a load ratio for each load component. These ratios are nondimensional coefficients used to denote the fraction of the allowable load that is developed under the combined loading system.

The load ratio (R) is defined as: $R = \frac{\text{Applied load}}{\text{Allowable load}}$

The margin of safety for each load component applied individually is given by the standard equation:

$$M.S. = \frac{1}{R} - 1$$

The effects of applying two or more loads simultaneously are displayed by combining the load ratios in an interaction equation or curve. Interaction equations are generally of the form:

$$\overline{R} = R_1^x + R_2^y + R_3^z + \ldots$$

where:

R_1, R_2, R_3 = load ratios of various load components or types (tension, bending, torsion, etc.);

x, y, z = exponents defining the interaction relationship.

Yielding or failure is indicated when the sum of these terms (R) equals 1.0.

The margin of safety of a structural member subjected to combined loads is:

$$M.S. = \frac{1}{R_{comb}} - 1$$

where $\frac{1}{R_{comb}}$ is the ratio by which all the load components can be increased to cause failure. R_{comb} is obtained by solving the following interaction equation:

$$\left(\frac{R_1}{R_{comb}}\right)^x + \left(\frac{R_2}{R_{comb}}\right)^y + \left(\frac{R_3}{R_{comb}}\right)^z + \cdots = 1.0$$

A graphical method is usually used to determine the margin of safety under combined loading. Figure 5.1 shows an example interaction curve for two simultaneously applied loads. The actual shape of the curve depends on the exponents x and y which, in turn, are dependent upon the geometry of the structure and the types of loading involved. Any particular curve represents all combinations of R1 and R2 that will cause yielding or failure.

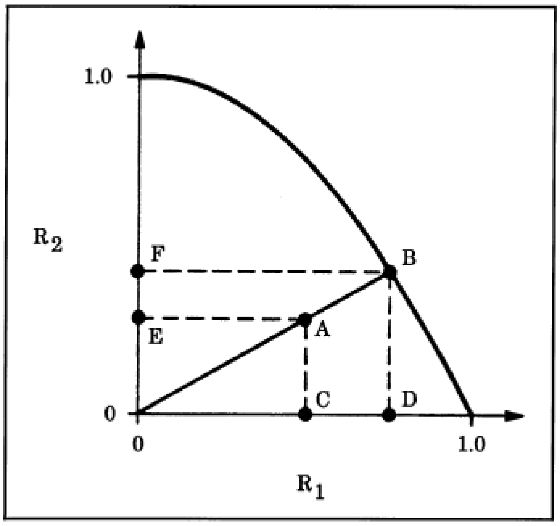


Figure 5.1. Interaction curve for two simultaneously applied loads.

Figures 5.2 & 5.3 present the symmetrical and unsymmetrical forms of the general interaction equation for two simultaneously applied loads. The appropriate curve or exponent to use for a particular structure and loading configuration should be obtained from the design manual appropriate to that condition.

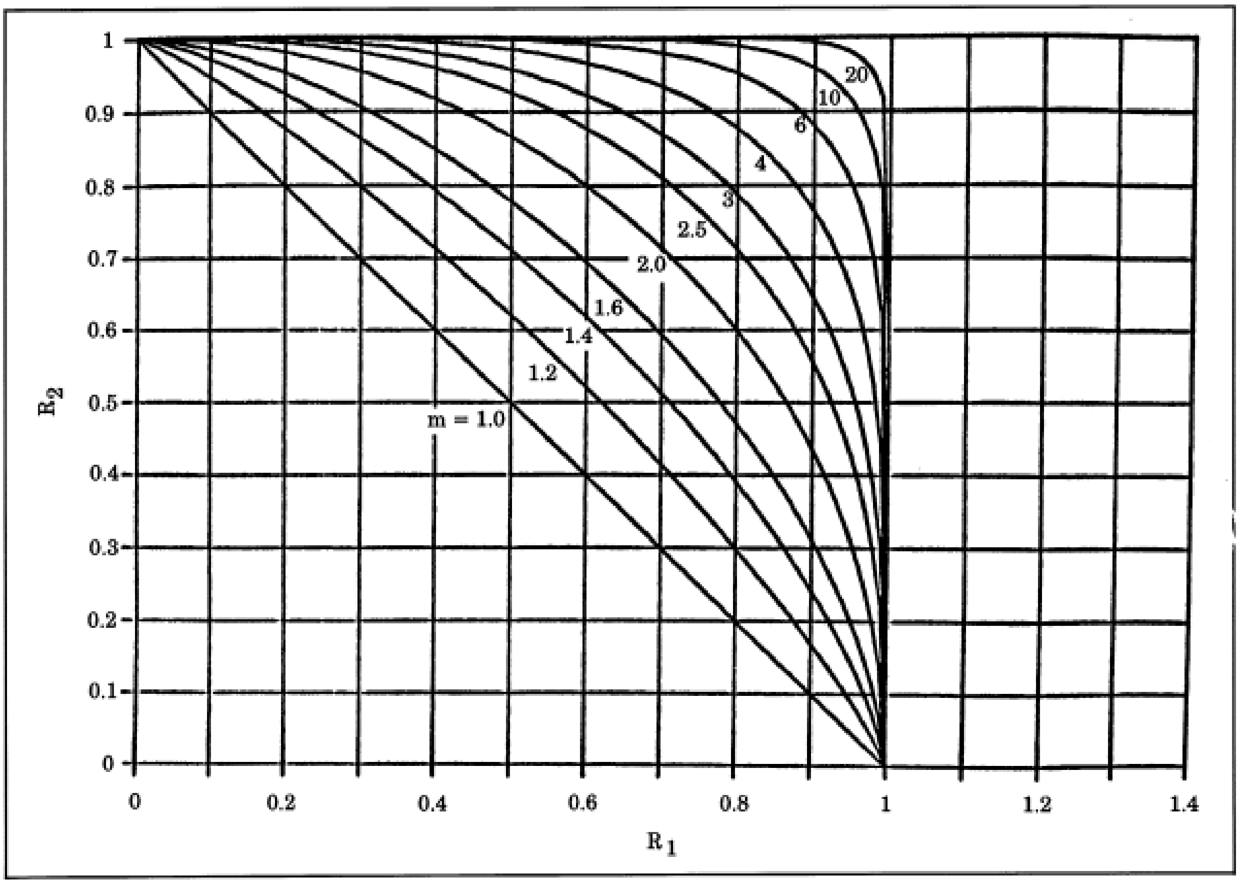


Figure 5.2. Curves for symmetrical form of the general interaction equation.

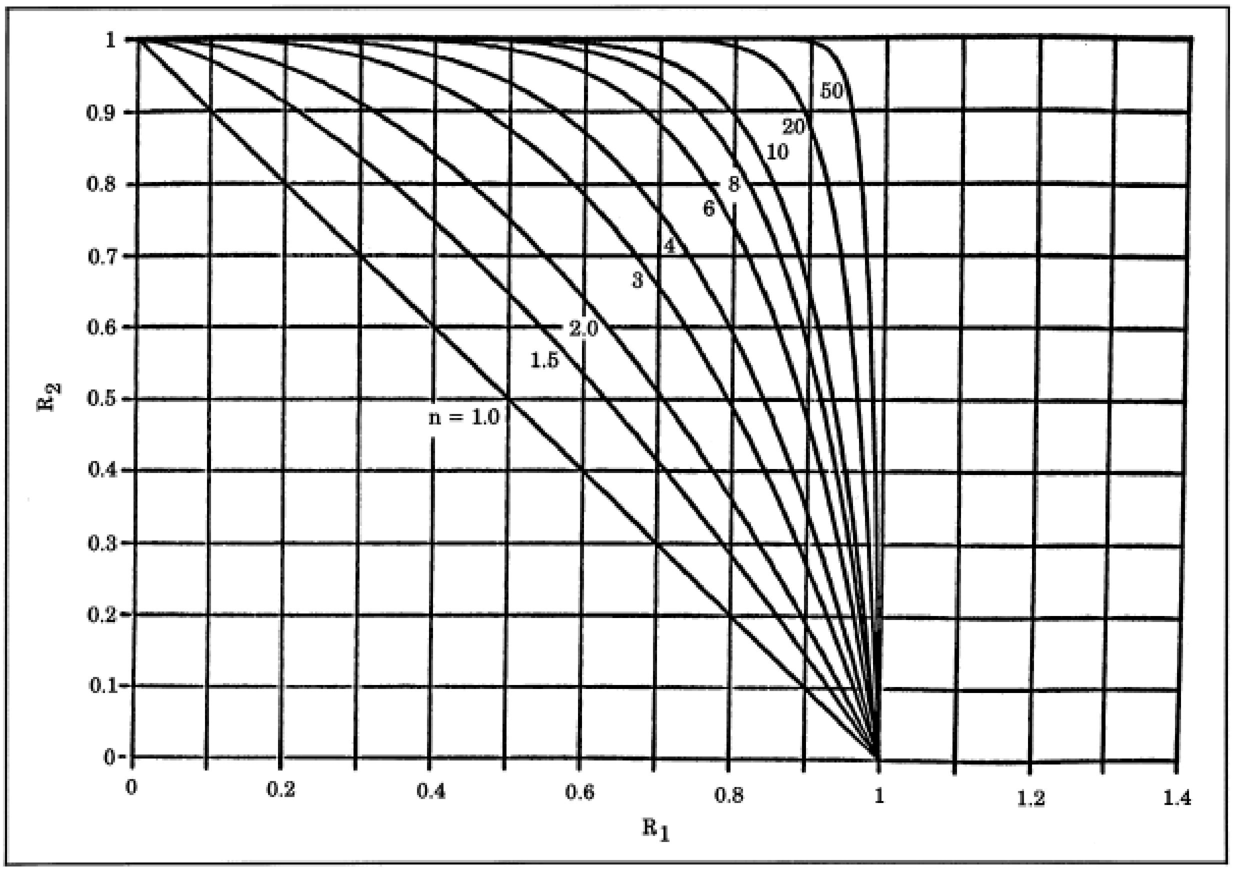


Figure 5.3. Curves for unsymmetrical form of the general interaction equation.

Floor Panel

The Floor Panel consist of two face shits (top and bottom) and a core between them (Figure 5.4).

Face Sheets: previous option – 2 ply (both sides) Phenolic Fiberglass Prepreg Fabric, then were replaced by Aluminum Alloy 2024.

Core: 0.95 in thick, 1/8 in Cell Nomex Honeycomb.

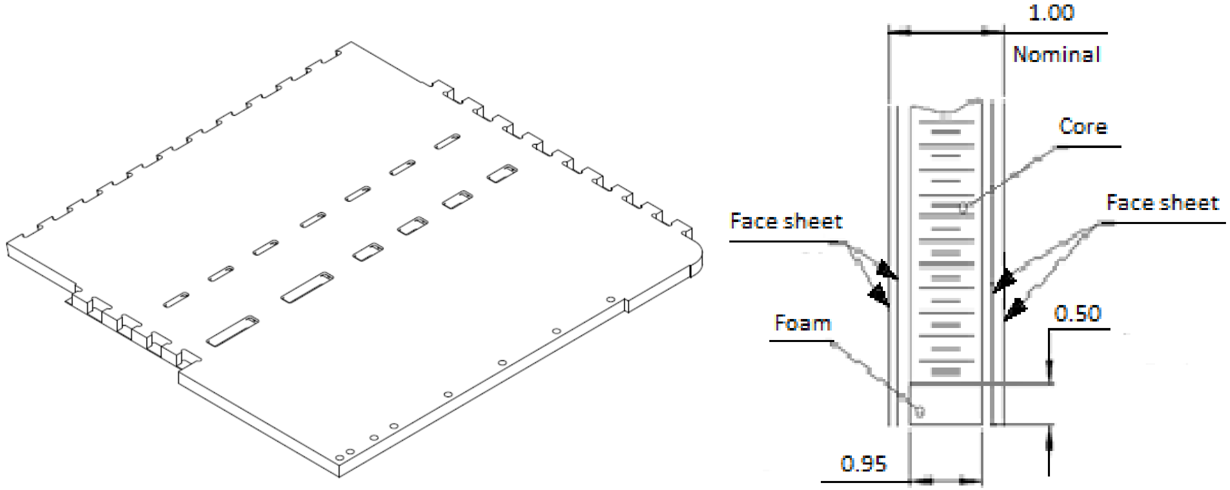


Figure 5.4. Floor Panel structure.

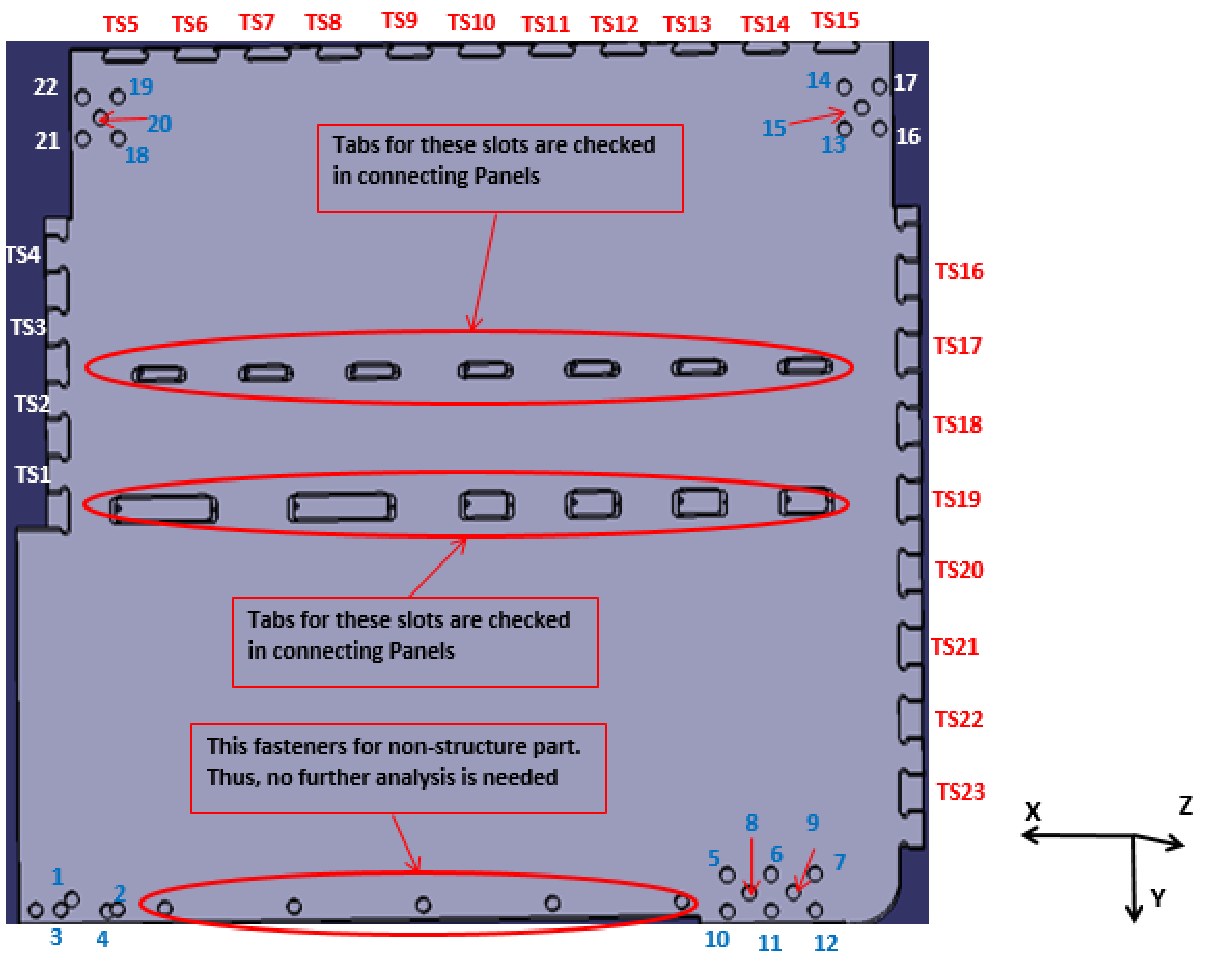


Figure 5.5. Numbering of Tabs/Slots and inserts of Floor Panel.

The Floor Panel contains 23 tabs/slots and 22 inserts. Each of them was analyzed, and margin of safety was obtained for each of them.

Different tabs/slots and inserts have various directions for tension load and shear load (Table 5.1).

Tab/Slot (or insert) number	Shear Plane	Tension Direction
TS1-TS4 & TS16-TS23	**Z-Y**	**X**
TS5-TS15	**X-Z**	**Y**
1-22	**X-Y**	**Z**

Table 5.1. Loads directions in Floor Panel.

The loads were extracted from finite element model for each tab/slot and insert. Next they were multiplied by the applicable load factors (for inertial loads in FEM, factor of 1 was used): 9G Forward, 6G Down, 3G Side and (6.2G Down 0.5G Fwd). The most critical load case was selected for further analysis.

Each of tab/slot and insert is numbered in the Figure 5.5, and the appropriate critical loads are presented in the Tables 5.2, 5.6.

Tab/Slot	Tension Load	PerpShear Load	ParaShear Load	Bending Moment
	Pt [lb]	Ps [lb]	Psi [lb]	M [lb*in]
TS1	19	51	101	33
TS2	48	67	31	53
TS3	56	41	18	53
TS4	45	49	20	47
TS5	15	123	219	19
TS6	6	15	158	4
TS7	1	35	169	1
TS8	9	23	143	8
TS9	9	6	152	7
TS10	1	4	155	3
TS11	6	5	162	7
TS12	9	2	173	12
TS13	10	7	190	11
TS14	3	35	209	5
TS15	6	109	218	14
TS16	26	1	59	57
TS17	60	8	12	62
TS18	53	24	11	58
TS19	53	38	19	52
TS20	14	12	53	59
TS21	6	9	61	67
TS22	2	19	79	67
TS23	16	69	111	53

Table 5.2. Floor Panel. Applied loads.

The maximum allowable loads were taken from the relevant BOEING regulatory documents (Table 5.3, 5.6, 5.7).

Tab/Slot	Tension Load	PerpShear Load	ParaShear Load	Bending Moment
	Pt_max	Ps_max	Psi_max	M_max
All	299	299	1004	480

Table 5.3. Floor Panel. Allowable loads.

Since the tabs/slots under analysis are subjected to combined load, a load ratio Ri was calculated for each type of loading:

- for tension load: Rt = Pt / Pt_max;
- for perpendicular shear load: Rs = Ps / Ps_max;
- for parallel shear load: Rsi = Psi / Psi_max;
- for bending moment: Rb = M / M_max.

All obtained Ri are listed in Table 5.4.

Tab/Slot	Rt	Rs	Rsi	Rb
TS1	0.06	0.17	0.10	0.07
TS2	0.16	0.23	0.03	0.11
TS3	0.19	0.14	0.02	0.11
TS4	0.15	0.17	0.02	0.10
TS5	0.05	0.41	0.22	0.04
TS6	0.02	0.05	0.16	0.01
TS7	0.00	0.12	0.17	0.00
TS8	0.03	0.08	0.14	0.02
TS9	0.03	0.02	0.15	0.01
TS10	0.00	0.01	0.15	0.01
TS11	0.02	0.02	0.16	0.02
TS12	0.03	0.01	0.17	0.02
TS13	0.03	0.02	0.19	0.02
TS14	0.01	0.12	0.21	0.01
TS15	0.02	0.36	0.22	0.03
TS16	0.09	0.00	0.06	0.12
TS17	0.20	0.03	0.01	0.13
TS18	0.18	0.08	0.01	0.12
TS19	0.18	0.13	0.02	0.11
TS20	0.05	0.04	0.05	0.12
TS21	0.02	0.03	0.06	0.14
TS22	0.01	0.06	0.08	0.14
TS23	0.05	0.23	0.11	0.11

Table 5.4. Floor Panel. Load ratio.

Margins of Safety were calculated for three types of loads combinations:

- Tension & Shear: MS Interaction is:

$$Rt+Rs=1$$

$$\Downarrow$$

$$MS = \frac{1}{(R_s + R_t)} - 1$$

- Bending & Tension: MS Interaction is:

$$Rt+Rb^{1.5}=1$$

$$\Downarrow$$

$$MS = \frac{1}{(R_s + R_b{}^{1.5})^{2/3}} - 1$$

- Bending & Shear: MS Interaction is:

$$Rb+Rs^{2}=1$$

$$\Downarrow$$

$$MS = \frac{1}{(R_s + R_b^2)^{1/2}} - 1$$

MS of each tab/slot for each type of loads combinations is shown in the Table_5.5.

Tab/Slot	Tension & Shear	Bending & Tension	Bending & Shear
TS1	+3.24	+4.27	+2.08
TS2	+1.59	+1.94	+1.48
TS3	+2.09	+1.71	+1.78
TS4	+2.16	+2.12	+1.83
TS5	+1.17	+5.62	**+1.13**
TS6	+4.64	Large	+1.52
TS7	+4.80	Large	+1.44
TS8	+4.76	+8.72	+1.64
TS9	+4.47	+8.70	+1.57
TS10	+5.26	Large	+1.54
TS11	+4.55	Large	+1.49
TS12	+3.91	+8.35	+1.40

TS13	+3.52	+8.25	+1.30
TS14	+3.61	Large	+1.19
TS15	+1.61	Large	+1.14
TS16	+5.81	+2.92	+1.90
TS17	+3.40	+1.55	+1.78
TS18	+2.86	+1.73	+1.79
TS19	+2.26	+1.79	+1.83
TS20	+8.95	+3.93	+1.83
TS21	Large	+4.78	+1.67
TS22	Large	+5.75	+1.65
TS23	+2.51	+4.00	+1.47

Table 5.5. Floor Panel. Margins of Safety.

All Margins of Safety of Tabs/Slots are positive, safety is provided.

Minimum MS is **+1.13** for TS5. Critical Load Case – 9G Forward. Mode of Failure is Ultimate Combined Load: Bending & Shear.

Next, insert check was performed (Table 5.6).

Insert	Applied Load		Edge distance [in]	Allowable Load [lb]		Rt	Rs	MS
	Papp [lb]	Vapp [lb]		Pmax	Vmax	Papp/Pmax	Vapp/Vmax	
1	26	279	0.5	299	299	0.086	0.934	**-0.02**
2	4	279	0.5	299	299	0.013	0.934	**+0.06**
3	7	185	0.5	299	299	0.023	0.618	**+0.56**
4	12	185	0.5	299	299	0.041	0.618	**+0.52**
5	There are connections of the Shear Plates with Panel. Thus, Shear Plates carry shear loads only.	278	1.5	299	299	0.000	0.929	**+0.08**
6		81	1.5	299	299	0.000	0.270	**+2.71**
7		126	1.5	299	299	0.000	0.420	**+1.38**
8		151	0.5	299	299	0.000	0.503	**+0.99**
9		77	0.5	299	299	0.000	0.258	**+2.87**
10		319	0.5	299	299	0.000	1.067	**-0.06**
11		73	0.5	299	299	0.000	0.245	**+3.09**
12		126	0.5	299	299	0.000	0.422	**+1.37**
13		265	1.5	299	299	0.000	0.886	**+0.13**
14		282	1.5	299	299	0.000	0.944	**+0.06**
15		183	0.5	299	299	0.000	0.613	**+0.63**
16		224	0.5	299	299	0.000	0.748	**+0.34**
17		243	0.5	299	299	0.000	0.814	**+0.23**
18		282	1.5	299	299	0.000	0.943	**+0.06**
19		292	1.5	299	299	0.000	0.976	**+0.02**
20		190	0.5	299	299	0.000	0.635	**+0.58**
21		231	0.5	299	299	0.000	0.773	**+0.29**
22		247	0.5	299	299	0.000	0.825	**+0.21**

Table 5.6. Floor Panel. Initial insert data.

The inserts under analysis are also subjected to combined load, a load ratio Ri was calculated for each type of loading:

- for in-plane shear load: Rs = VApplied / VMax;
- for tension load Rt = PApplied / PMax.

Margins of Safety was calculated for Tension/Shear Interaction:

$$MS = \frac{1}{(R_T + R_S)} - 1$$

Several negative MS were obtained: insert №1, 10 (Table 5.6).

In order to increase these Margins of Safety to obtain positive values for them, it was decided to replace the face sheet material with the aluminum alloy 2024. This alloy has larger allowable loads compared to fiberglass and therefore the Margins of Safety will also be greater (Table 5.7).

Insert	Applied Load		Edge distance [in]	Allowable Load [lb]		Rt	Rs	MS
	Papp [lb]	Vapp [lb]		Pmax	Vmax	Papp/Pmax	Vapp/Vmax	
1	26	279	0.5	328	325	0.078	0.859	**+0.07**
2	4	279	0.5	328	325	0.012	0.859	**+0.15**
3	7	185	0.5	328	325	0.021	0.568	**+0.70**
4	12	185	0.5	328	325	0.037	0.568	**+0.65**
5	There are connections of the Shear Plates with Panel. Thus, Shear Plates carry shear loads only.	278	1.5	328	553	0.000	0.503	**+0.99**
6		81	1.5	328	553	0.000	0.146	**+5.86**
7		126	1.5	328	553	0.000	0.227	**+3.40**
8		151	0.5	328	325	0.000	0.463	**+1.16**
9		77	0.5	328	325	0.000	0.238	**+3.21**
10		319	0.5	328	325	0.000	0.981	**+0.02**
11		73	0.5	328	325	0.000	0.225	**+3.44**
12		126	0.5	328	325	0.000	0.388	**+1.58**
13		265	1.5	328	553	0.000	0.479	**+1.09**
14		282	1.5	328	553	0.000	0.511	**+0.96**
15		183	0.5	328	325	0.000	0.564	**+0.77**
16		224	0.5	328	325	0.000	0.688	**+0.45**
17		243	0.5	328	325	0.000	0.749	**+0.33**
18		282	1.5	328	553	0.000	0.510	**+0.96**
19		292	1.5	328	553	0.000	0.528	**+0.90**
20		190	0.5	328	325	0.000	0.584	**+0.71**
21		231	0.5	328	325	0.000	0.711	**+0.41**
22		247	0.5	328	325	0.000	0.759	**+0.32**

Table 5.7. Floor Panel. Updated insert data.

After replacing the material of the face sheets, all insert's Margins of Safety are positive, safety is provided.

Minimum MS is **+0.02** for insert №10. Critical Load Case – 9G Forward. Mode of Failure is Ultimate Shear Load.

Calculation for Tabs/Slots left unchanged, conservatively.

AFT Panel

The AFT Panel consist of two face shits (top and bottom) and a core between them (Figure 5.6).

Face Sheets: 2 ply (both sides) Phenolic Fiberglass Prepreg Fabric.

Core: 0.95 in thick, 1/8 in Cell Nomex Honeycomb.

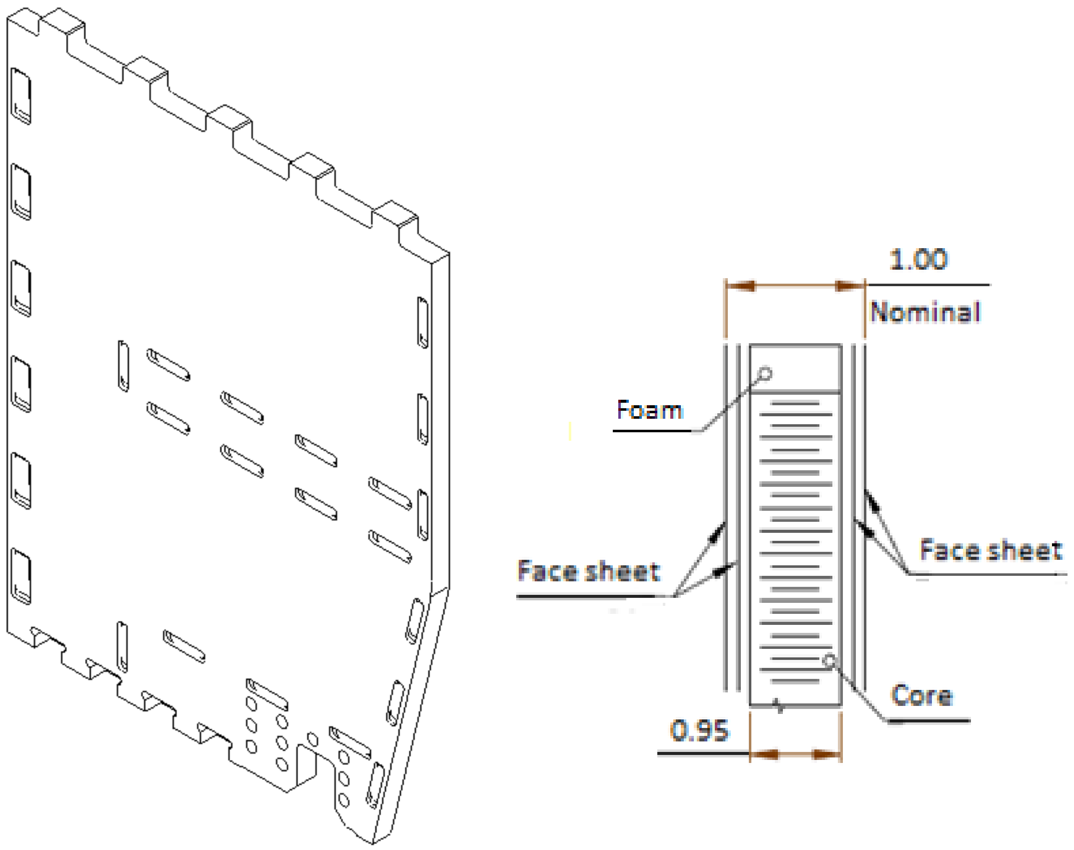


Figure 5.6. AFT Panel structure.

To analyze the AFT Panel, the same method was used as for the Floor Panel. Data of analysis results are given in the following section.

Tab/Slot (or insert) number	Shear Plane	Tension Direction
TS1-TS9	**X-Y**	**Z**
1-10	**Y-Z**	**X**

Table 5.8. Loads directions in AFT Panel.

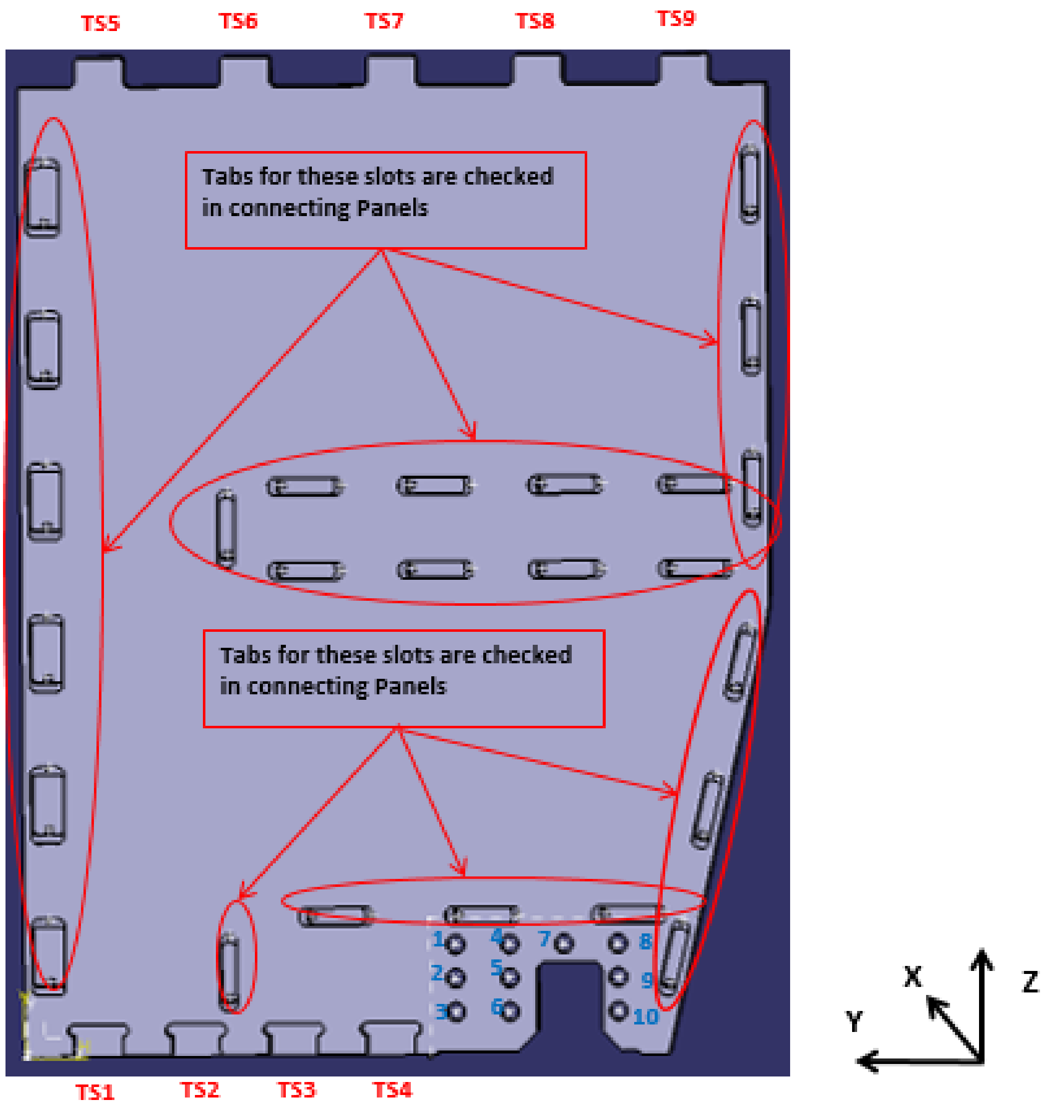


Figure 5.7. Numbering of Tabs/Slots and inserts of AFT Panel.

Tab/Slot	Tension Load	PerpShear Load	ParaShear Load	Bending Moment
	Pt [lb]	Ps [lb]	Psi [lb]	M [lb*in]
TS1	51	19	101	33
TS2	67	48	31	53
TS3	41	56	18	53
TS4	49	45	20	47
TS5	73	25	41	60
TS6	96	3	47	145
TS7	163	20	77	182
TS8	80	9	79	115
TS9	172	7	20	127

Table 5.9. AFT Panel. Applied loads.

Tab/Slot	Tension Load	PerpShear Load	ParaShear Load	Bending Moment
	Pt_max	Ps_max	Psi_max	M_max
TS1-TS4	299	299	1004	480
TS5-TS9	380	355	978	660

Table 5.10. AFT Panel. Allowable loads.

Tab/Slot	Rt	Rs	Rsi	Rb
TS1	0.17	0.06	0.10	0.07
TS2	0.23	0.16	0.03	0.11
TS3	0.14	0.19	0.02	0.11
TS4	0.17	0.15	0.02	0.10
TS5	0.19	0.07	0.04	0.09
TS6	0.25	0.01	0.05	0.22
TS7	0.43	0.06	0.08	0.28
TS8	0.21	0.02	0.08	0.17
TS9	0.45	0.02	0.02	0.19

Table 5.11. AFT Panel. Load ratio.

Tab/Slot	Tension & Shear	Bending & Tension	Bending & Shear
TS1	+2.68	+2.03	+2.08
TS2	+1.59	+1.44	+1.70
TS3	+2.09	+2.22	+1.62
TS4	+2.16	+1.97	+1.88
TS5	+2.83	+1.76	+2.24
TS6	+2.32	+0.99	+1.14
TS7	+0.97	**+0.45**	+0.89
TS8	+2.44	+1.32	+1.39
TS9	+1.12	+0.52	+1.28

Table 5.12. AFT Panel. Margins of Safety.

All Margins of Safety of Tabs/Slots are positive, safety is provided.

Minimum MS is +0.45 for TS7. Critical Load Case – 9G Forward. Mode of Failure is Ultimate Combined Load: Bending & Tension.

Insert	Applied Load		Edge distance [in]	Allowable Load [lb]		Rt	Rs	MS
	Papp [lb]	Vapp [lb]		Pmax	Vmax	Papp/Pmax	Vapp/Vmax	
1	6	260	1.5	344	305	0.016	0.853	**+0.15**
2	0	138	1.5	344	305	0.000	0.451	**+1.21**
3	32	97	0.5	344	305	0.094	0.317	**+1.43**
4	6	294	0.5	344	305	0.017	0.965	**+0.02**
5	5	140	0.5	344	305	0.015	0.458	**+1.11**
6	32	127	0.5	344	305	0.093	0.415	**+0.97**
7	1	368	0.5	344	305	0.002	1.206	**-0.17**
8	5	659	0.5	344	305	0.014	2.160	**-0.54**
9	7	370	0.5	344	305	0.020	1.212	**-0.19**
10	25	330	0.5	344	305	0.074	1.080	**-0.13**
Total Shear Load		2781						

Table 5.13. AFT Panel. Initial insert data.

Several negative MS were obtained: insert №7-10 (Table 5.13).

To solve this problem, it was decided to add foam instead of core material in the area of mounting of inserts, since foam has larger values of allowable loads compared to Nomex (Table 5.14).

Insert	Applied Load		Edge distance [in]	Allowable Load [lb]		Rt	Rs	MS
	Papp [lb]	Vapp [lb]		Pmax	Vmax	Papp/Pmax	Vapp/Vmax	
1	6	260	1.5	449	385	0.012	0.676	**+0.45**
2	0	138	1.5	449	385	0.000	0.357	**+1.80**
3	32	97	0.5	449	385	0.072	0.251	**+2.10**
4	6	294	0.5	449	385	0.013	0.764	**+0.29**
5	5	140	0.5	449	385	0.011	0.363	**+1.67**
6	32	127	0.5	449	385	0.072	0.329	**+1.50**
7	1	368	0.5	449	385	0.001	0.955	**+0.05**
8	5	659	0.5	449	385	0.011	1.711	**-0.42**
9	7	370	0.5	449	385	0.015	0.960	**+0.02**
10	25	330	0.5	449	385	0.057	0.856	**+0.10**

Table 5.14. AFT Panel. Updated insert data.

After adding foam, Margins of Safety for inserts №7, 9, 10 became positive. However, for Insert №8 this was not enough, there was still negative MS for it. To correct it, Adhesive Bond was used for connecting Fitting with Panel (Figure 5.8). The Adhesive Bond added Shear capability.

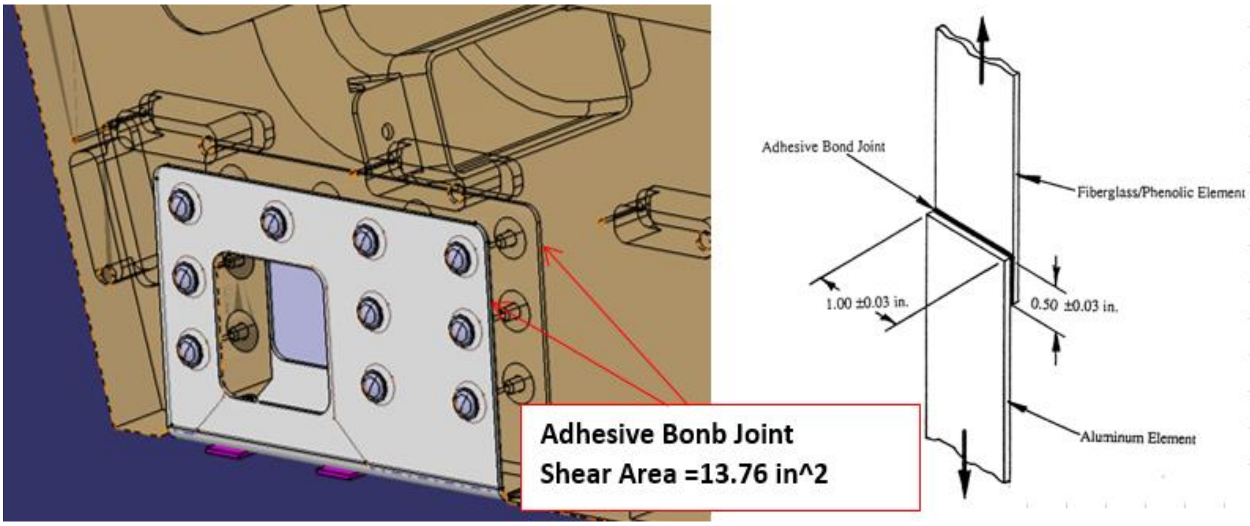


Figure 5.8. Adhesive Bond for connecting Fitting with Panel.

Additional calculation was performed for Adhesive Bond:

- applied load:

$$P_{app} = \text{Total Shear Load} / 2 = 2781/2 = 1391 \text{ lb};$$

- allowable load (according to the BOEING regulatory document, P_{all} per 1 square inch equals 1020 psi):

$$P_{all} = 1020*13.76 = 14035 \text{ lb};$$

- Margin of Safety:

$$MS = P_{all} / P_{app} - 1 = 14035/1391 - 1 = \mathbf{+9.09.}$$

Insert №8 is critical. Critical Load Case – 9G Forward. Mode of Failure is Ultimate Combined Load: Tension & Shear. For insert №8, Margin of Safety is negative, but since Adhesive Bond was added at critical place and Margin of Safety of Adhesive Bond is positive (**+9.09**), the calculation is considered acceptable and safety is ensured.

Lower Outboard Panel

The Lower Outboard Panel consist of two face shits (top and bottom) and a core between them (Figure 5.9).

Face Sheets: 2 ply (both sides) Phenolic Fiberglass Prepreg Fabric.

Core: 0.47 in thick, 1/8 in Cell Nomex Honeycomb.

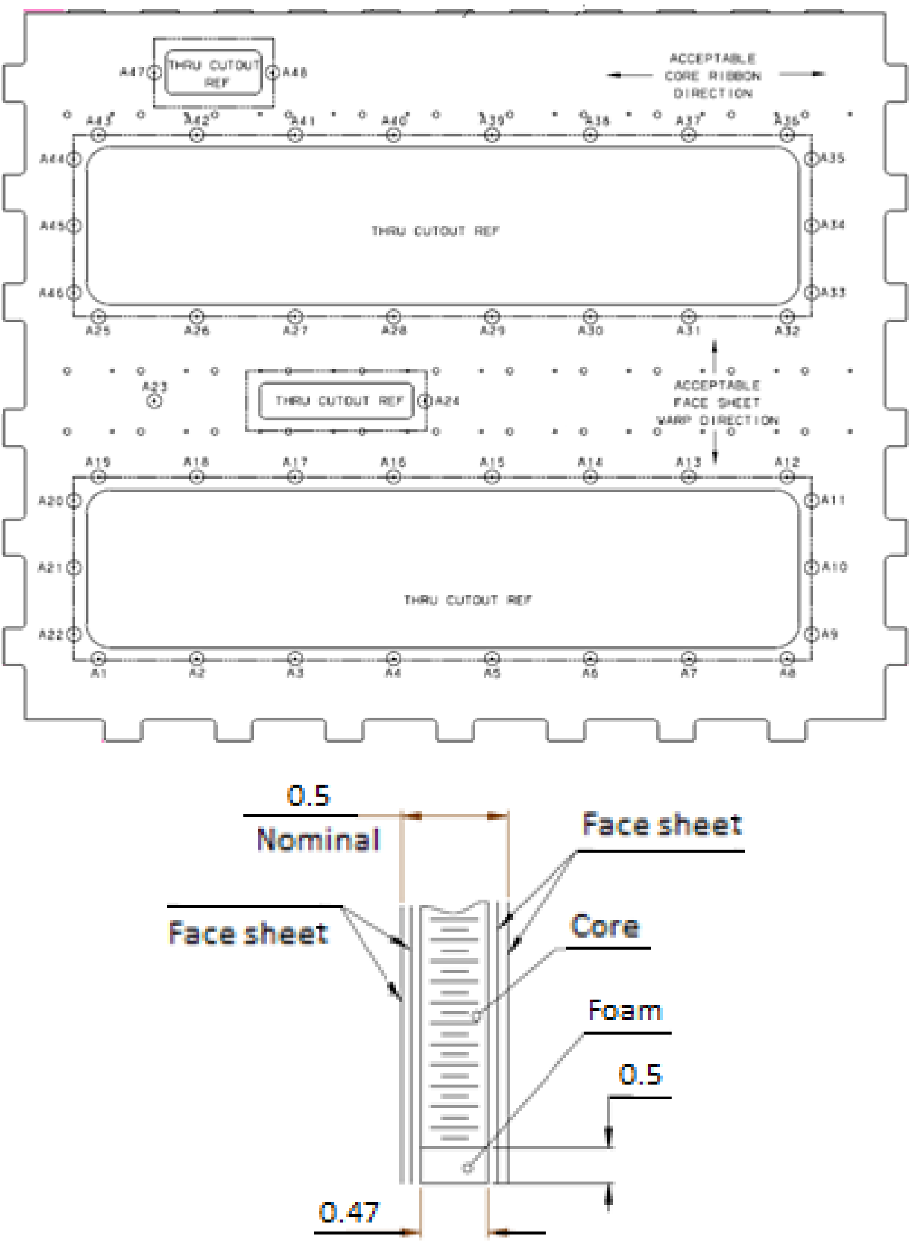


Figure 5.9. Lower Outboard Panel structure.

Analysis of the Lower Outboard Panel is similar to the previous. Data of analysis results are given in this section.

To get more accurate calculation result and to avoid material overruns, core ribbon direction and face sheet warp direction were taken into account.

Tab-Slot (or insert) number	Shear Plane	Tension Direction
TS1-TS11 & TS18-TS24	**X-Y**	**Z**
TS12-TS17 & TS25-TS30	**Z-Y**	**X**
1- 44	**X-Z**	**Y**

Table 5.15. Loads directions in Lower Outboard Panel.

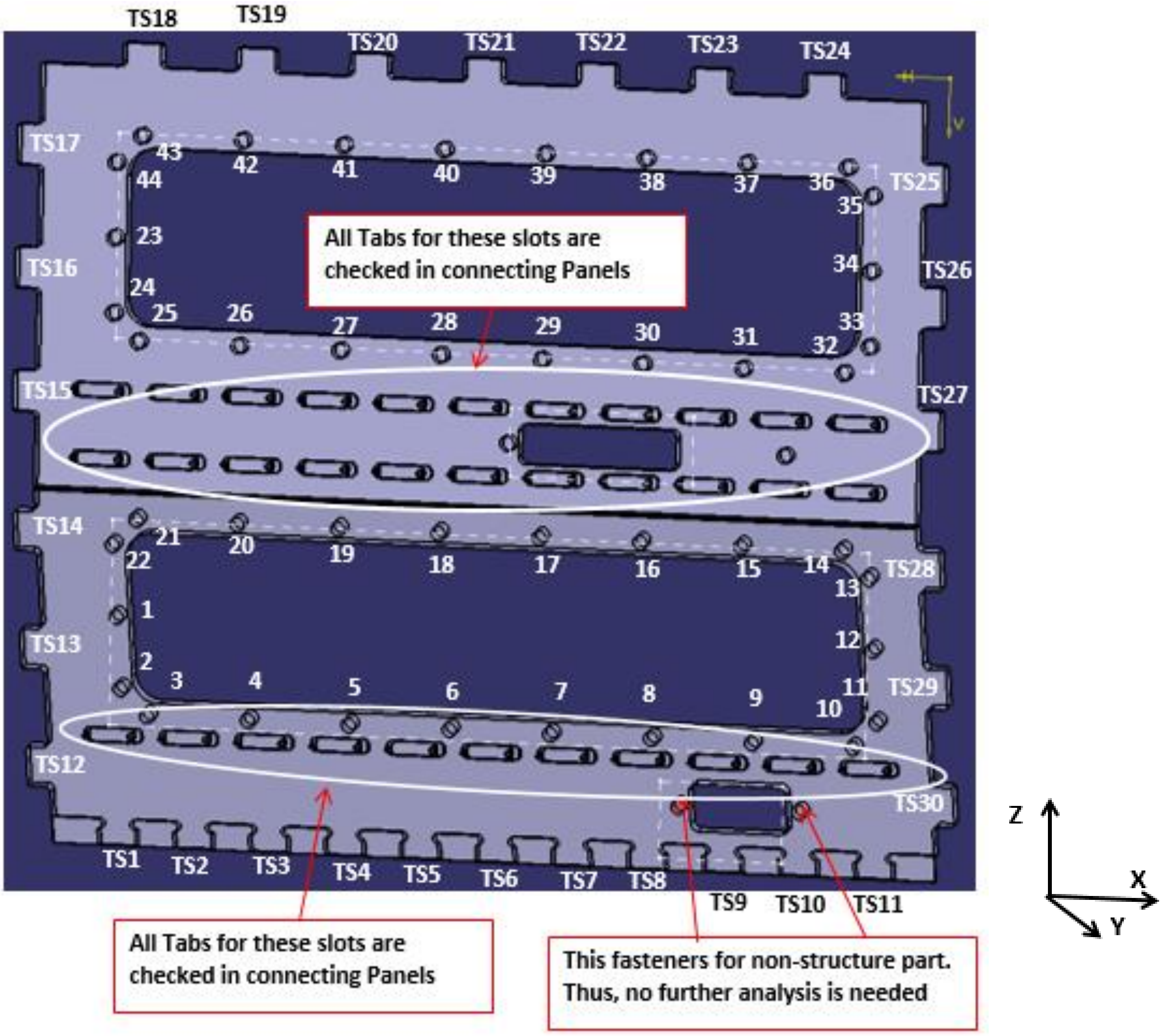


Figure 5.10. Numbering of Tabs/Slots and inserts of Lower Outboard Panel.

Tab/Slot	Tension Load	PerpShear Load	ParaShear Load	Bending Moment
	Pt [lb]	Ps [lb]	Psi [lb]	M [lb*in]
TS1	104	21	219	14
TS2	31	11	209	5
TS3	3	11	190	11
TS4	7	8	173	12
TS5	8	4	162	7
TS6	5	2	155	3
TS7	5	11	152	7
TS8	22	14	143	8
TS9	36	7	169	1
TS10	15	10	158	4
TS11	125	15	217	19

TS12	32	24	264	5
TS13	9	1	246	13
TS14	2	26	208	13
TS15	2	2	181	2
TS16	1	1	170	3
TS17	24	11	128	6
TS18	102	3	159	30
TS19	27	4	141	6
TS20	7	1	126	1
TS21	0	0	122	2
TS22	7	1	123	3
TS23	32	4	133	9
TS24	156	5	178	51
TS25	23	5	128	9
TS26	6	1	179	6
TS27	3	3	185	3
TS28	8	36	209	9
TS29	14	0	240	8
TS30	29	23	269	3

Table 5.16. Lower Outboard Panel Panel. Applied loads.

Tab/Slot	Tension Load	PerpShear Load	ParaShear Load	Bending Moment
	Pt_max	Ps_max	Psi_max	M_max
All	369	247	892	246

Table 5.17. Lower Outboard Panel. Allowable loads.

Tab/Slot	Rt	Rs	Rsi	Rb
TS1	0.28	0.08	0.25	0.06
TS2	0.08	0.04	0.23	0.02
TS3	0.01	0.04	0.21	0.05
TS4	0.02	0.03	0.19	0.05
TS5	0.02	0.02	0.18	0.03
TS6	0.01	0.01	0.17	0.01
TS7	0.01	0.04	0.17	0.03
TS8	0.06	0.06	0.16	0.03
TS9	0.10	0.03	0.19	0.01
TS10	0.04	0.04	0.18	0.02
TS11	0.34	0.06	0.24	0.08
TS12	0.09	0.10	0.30	0.02
TS13	0.02	0.00	0.28	0.05
TS14	0.01	0.11	0.23	0.05
TS15	0.01	0.01	0.20	0.01
TS16	0.00	0.00	0.19	0.01
TS17	0.07	0.04	0.14	0.03
TS18	0.28	0.01	0.18	0.12
TS19	0.07	0.01	0.16	0.03

TS20	0.02	0.01	0.14	0.00
TS21	0.00	0.00	0.14	0.01
TS22	0.02	0.01	0.14	0.01
TS23	0.09	0.02	0.15	0.04
TS24	0.42	0.02	0.20	0.21
TS25	0.06	0.02	0.14	0.04
TS26	0.02	0.00	0.20	0.03
TS27	0.01	0.01	0.21	0.01
TS28	0.02	0.15	0.23	0.03
TS29	0.04	0.00	0.27	0.03
TS30	0.08	0.09	0.30	0.01

Table 5.18. Lower Outboard Panel. Load ratio.

Tab/Slot	Tension & Shear	Bending & Tension	Bending & Shear
TS1	+0.89	+1.25	+1.00
TS2	+2.13	+4.08	+1.06
TS3	+3.54	Large	+1.16
TS4	+3.71	+9.77	+1.26
TS5	+3.94	Large	+1.34
TS6	+4.31	Large	+1.40
TS7	+4.42	Large	+1.42
TS8	+3.55	+5.16	+1.49
TS9	+2.48	+3.69	+1.30
TS10	+3.57	+7.01	+1.38
TS11	**+0.72**	+0.97	+1.00
TS12	+1.62	+4.02	+0.84
TS13	+2.34	+8.11	+0.90
TS14	+3.17	Large	+1.06
TS15	+3.79	Large	+1.22
TS16	+4.18	Large	+1.29
TS17	+3.78	+4.92	+1.63
TS18	+1.20	+1.15	+1.27
TS19	+3.36	+4.56	+1.51
TS20	+5.26	Large	+1.66
TS21	+6.26	Large	+1.70
TS22	+5.33	Large	+1.69
TS23	+3.21	+3.80	+1.57
TS24	+0.61	+0.56	+1.03
TS25	+3.89	+5.02	+1.63
TS26	+3.60	Large	+1.23
TS27	+3.61	Large	+1.19
TS28	+2.92	+9.99	+1.06
TS29	+2.27	+7.25	+0.92
TS30	+1.62	+4.33	+0.82

Table 5.19. Lower Outboard Panel. Margins of Safety.

All Margins of Safety of Tabs/Slots are positive, safety is provided.

Minimum MS is **+0.72** for TS11. Critical Load Case – 9G Forward. Mode of Failure is Ultimate Combined Load: Tension & Shear.

Insert	Applied Load		Edge distance [in]	Allowable Load [lb]		Rt	Rs	MS
	Papp [lb]	Vapp [lb]		Pmax	Vmax	Papp/Pmax	Vapp/Vmax	
1	1	74	0.5	118	310	0.011	0.240	**+2.98**
2	4	99	0.5	118	310	0.031	0.321	**+1.85**
3	1	138	0.5	118	310	0.012	0.444	**+1.19**
4	1	172	0.5	118	310	0.006	0.555	**+0.78**
5	1	191	0.5	118	310	0.006	0.617	**+0.61**
6	1	200	0.5	118	310	0.005	0.645	**+0.54**
7	0	194	0.5	118	310	0.003	0.626	**+0.59**
8	0	177	0.5	118	310	0.003	0.570	**+0.74**
9	1	169	0.5	118	310	0.009	0.545	**+0.81**
10	4	143	0.5	118	310	0.034	0.460	**+1.02**
11	3	99	0.5	118	310	0.026	0.318	**+1.91**
12	1	85	0.5	118	310	0.009	0.273	**+2.55**
13	11	124	0.5	118	310	0.091	0.401	**+1.03**
14	1	128	0.5	118	310	0.009	0.413	**+1.37**
15	1	148	0.5	118	310	0.005	0.477	**+1.08**
16	1	160	0.5	118	310	0.008	0.516	**+0.91**
17	1	162	0.5	118	310	0.011	0.522	**+0.88**
18	1	152	0.5	118	310	0.005	0.489	**+1.03**
19	1	154	0.5	118	310	0.007	0.497	**+0.99**
20	1	144	0.5	118	310	0.005	0.464	**+1.13**
21	0	137	0.5	118	310	0.002	0.441	**+1.25**
22	5	132	0.5	118	310	0.042	0.427	**+1.13**
23	42	2	0.5	118	310	0.356	0.008	**+1.75**
24	36	44	0.5	118	310	0.307	0.141	**+1.23**
25	34	68	0.5	118	310	0.292	0.218	**+0.96**
26	21	88	0.5	118	310	0.177	0.284	**+1.17**
27	16	96	0.5	118	310	0.139	0.311	**+1.22**
28	20	105	0.5	118	310	0.173	0.339	**+0.95**
29	21	104	0.5	118	310	0.178	0.335	**+0.95**
30	18	96	0.5	118	310	0.154	0.309	**+1.16**
31	30	80	0.5	118	310	0.257	0.257	**+0.95**
32	41	56	0.5	118	310	0.351	0.181	**+0.88**
33	45	35	0.5	118	310	0.383	0.112	**+1.02**
34	56	9	0.5	118	310	0.477	0.028	**+0.98**
35	44	59	0.5	118	310	0.369	0.190	**+0.79**
36	14	78	0.5	118	310	0.123	0.252	**+1.67**
37	20	82	0.5	118	310	0.170	0.266	**+1.30**
38	14	80	0.5	118	310	0.116	0.260	**+1.66**
39	4	82	0.5	118	310	0.030	0.264	**+2.40**
40	4	83	0.5	118	310	0.034	0.267	**+2.32**

41	9	82	0.5	118	310	0.078	0.263	**+1.93**
42	8	80	0.5	118	310	0.069	0.257	**+2.06**
43	13	70	0.5	118	310	0.111	0.227	**+1.96**
44	30	50	0.5	118	310	0.250	0.162	**+1.43**

Table 5.20. Lower Outboard Panel. Insert data.

All insert's Margins of Safety are positive, safety is provided.

Minimum MS is **+0.54** for insert №6. Critical Load Case – 9G Forward. Mode of Failure is Ultimate Combined Load: Tension & Shear.

Additional check was made for Lower Outboard Panel, as this panel is not flat. In the PWS structure in question, it was bent at an angle of 166.3 degrees. For this, groove was cut in the middle of the panel (Figure 5.11). This weakened the structure, and therefore this place should be checked additionally.

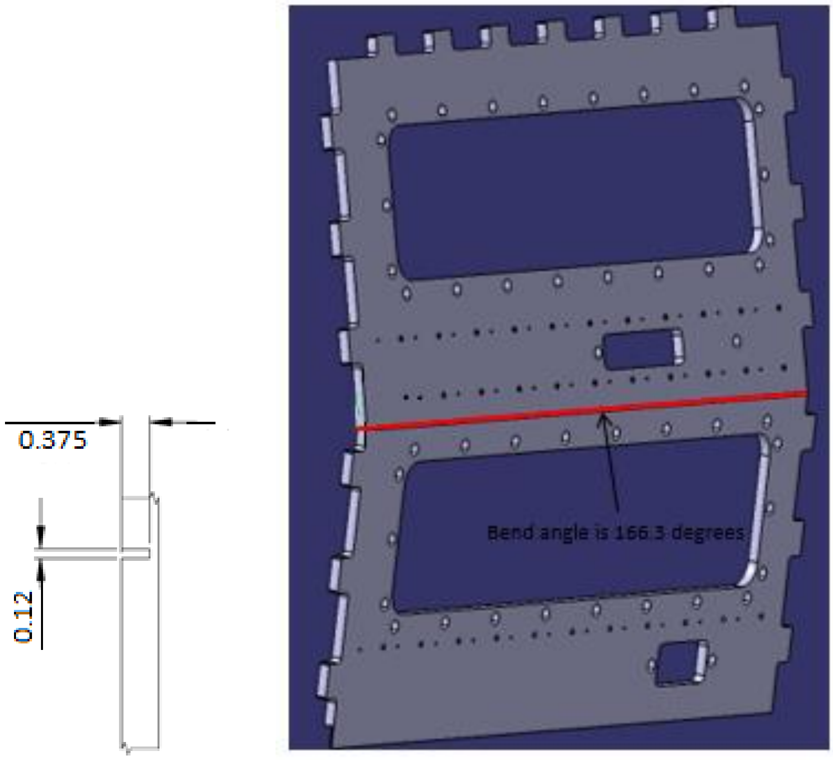


Figure 5.11. Location of bend of Lower Outboard Panel.

Groove loads were obtained from Patran (Figure 5.12).

By FEM the critical load case is 9G FWD:

1G FWD: Fx = 198.60 lb; Fy = 0.95 lb; Fz = -6.75 lb;

Mx = -4.39 lb*in; My = 847.84 lb*in; Mz = 82.16 lb*in.

Since the type of solution is linear, the loads can be adjusted with appropriate coefficients:

9G FWD: Fx = 1787.40 lb; Fy = 8.55 lb; Fz = -60.75 lb;

Mx = -39.51 lb*in; My = 7630.56 lb*in; Mz = 739.44 lb*in.

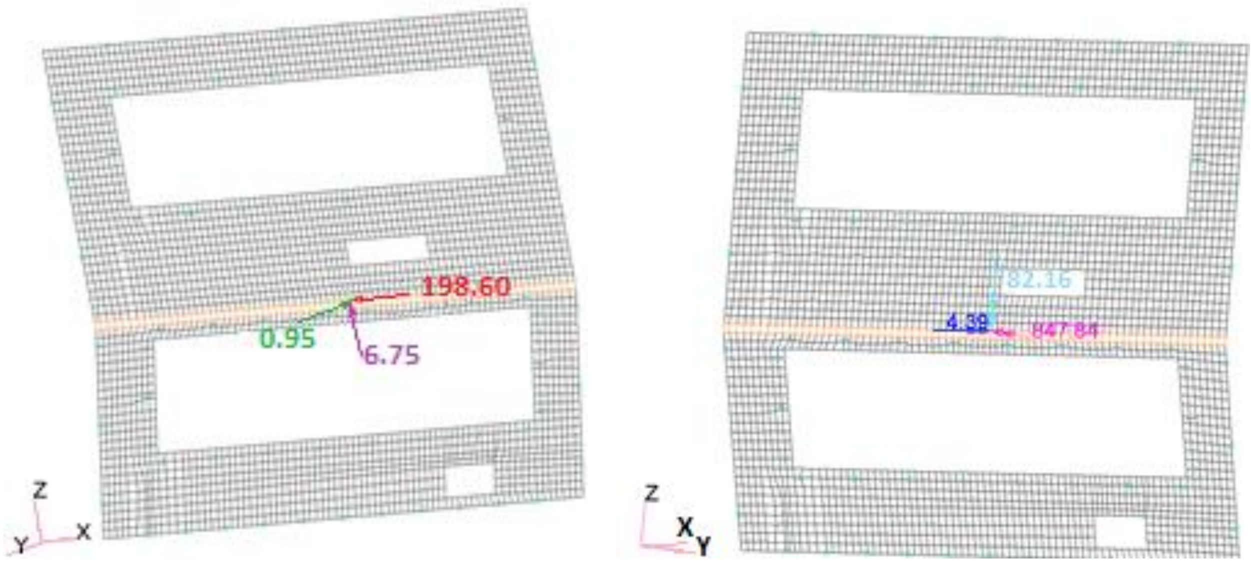


Figure 5.12. Loads in location of bend.

Applied shear flow at the panel joint is:

$$V = \frac{F_{SHEAR}}{L}.$$

L = 35 in; F_{shear} = 8.55 lb; V = 8.55/35 = 0.24 lb/in.

Allowable shear flow is:

Fs = 27.25 lb/in.

MS Calculation:

$$MS = \frac{Fs}{V} - 1.$$

MS = 27.25/0.24 – 1 = **Large**.

Applied Bending Moment:

L = 35 in; M_{bend} = 39.51 = in*lb;

$M_{applied}$ = M_{bend}/L = 39.51/35 = 1.13 in*lb/in.

Allowable Bending Moment:

M_{allow} = 19.5 in*lb/in.

MS Calculation:

$$MS = \frac{Mallow}{Mapplied} - 1.$$

MS = 19.5/1.13 – 1 = **Large**.

Also check of Flexural Strength of Lower Outboard Panel at sections A – A and B – B was carried out using Microsoft Excel.

By FEM the critical load case is 9G FWD.

Face Sheets

Mi = Mmax / b

Mmax = 9*Mx (extracted from FEM, 1g FWD)

Mmax = 72,5 in-lb

b = 2,50 in

Mi = 28,98 in-lb/in

d = h-[(t1+t2)/2]

t1 = t2 = 0,015 in

h = 0,50 in (panel thickness)

d = 0,485 in

f1 = f2 = Mi/dt1

f1 = f2 = 3984 psi

Fb = 14 ksi

$$MS = \left(\frac{F_b}{f_2}\right) - 1$$

MS = **+2,51**

Allowables 0.47" core:

Fult	14	ksi	face sheet tension/compression
Fcc	1400	psi	core compression
Fsu	416	psi	core Longitudinal shear

For Face Sheet bending

A – A

Core

V1 = Vmax / b

Vmax = 9*Vx (extracted from FEM, 1g FWD)

Vmax = 283,0 lb/in

V1 = 113,18 lb/in

c = Core Thickness

c = 0,470 in

fsc = V1/c

fsc = 241 psi

Fsw = 416 psi

$$MS = \left(\frac{F_{sw}}{f_{sc}}\right) - 1$$

MS = **+0,73** **For Core Shear**

$f_{cc} = M_{max}^2 / b^2 d D$

$D = E(I_i)$

$E_{cc} = 19000$ psi (core)

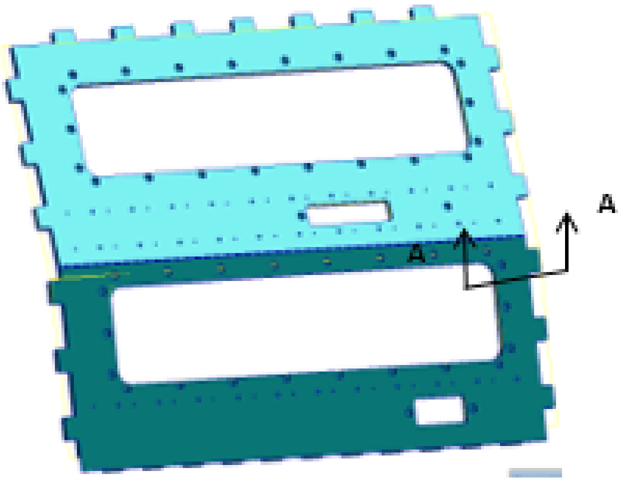
$I_i = d^2(t_1 t_2 / t_1 + t_2)$

$D = 34$ lb-in²/in

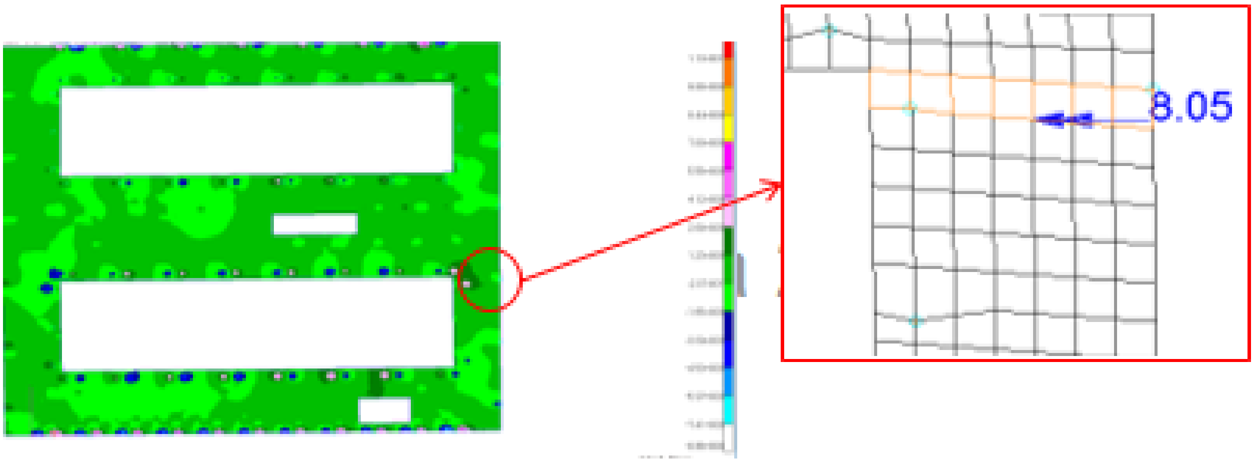
$f_{cc} = 52$ psi

$$MS = \left(\frac{F_{cc-core}}{f_{cc}} \right) - 1$$

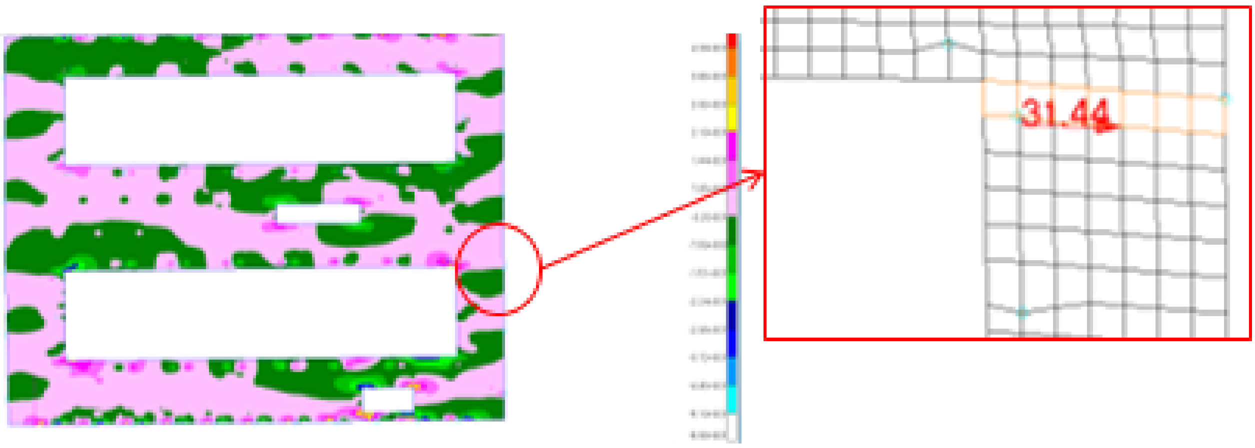
MS = Large Core Crushing



1G FWD, Moment Results, X Component



1G FWD, Forces Results, X Component



By FEM the critical load case is 9G FWD

Face Sheets

Mi = Mmax / b
Mmax = 9*Mx (extracted from FEM, 1g FWD)
Mmax = 67,5 in-lb
b = 3,50 in
Mi = 19,286 in-lb/in
d = h-[(t1+t2)/2]
t1 = t2 = 0,015 in
h = 0,50 in (panel thickness)
d = 0,485 in
f1 = f2 = Mi/dt1
f1 = f2 = 2651 psi
Fb = 14 ksi

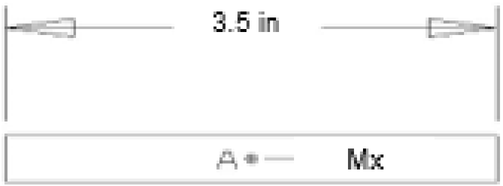
$$MS = \left(\frac{F_b}{f_2}\right) - 1$$

MS = **+4,28**

Allowables 0.47" core:

Fult	14	ksi	face sheet tension/compression
Fcc	1400	psi	core compression
Fsu	416	psi	core Longitudinal shear

For Face Sheet bending



B-B

Core

V1 = Vmax / b
Vmax = 9*Vx (extracted from FEM, 1g FWD)
Vmax = 350,9 lb/in
V1 = 100,26 lb/in
c = Core Thickness
c = 0,470 in
fsc = V1/c
fsc = 213 psi
Fsw = 416 psi

$$MS = \left(\frac{F_{sw}}{f_{sc}}\right) - 1$$

MS = **+0,95** Core Shear

$f_{cc} = M_{max}^2 / b^2 dD$

$D = E(I_i)$

$E_{cc} = 19000$ psi (core)

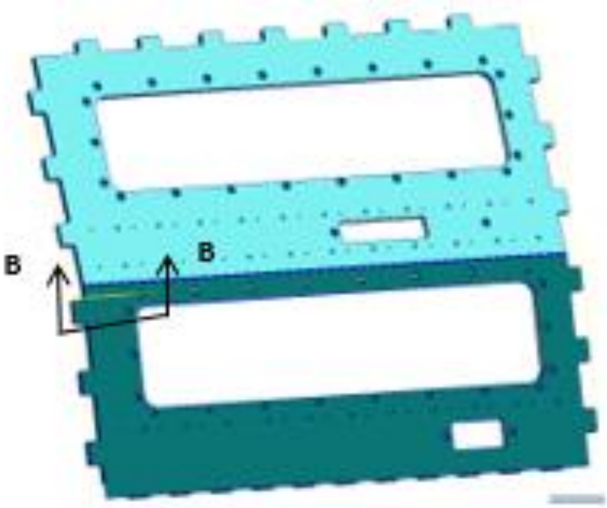
$I_i = d^2(t_1 t_2 / t_1 + t_2)$

$D = 34$ lb-in^2/in

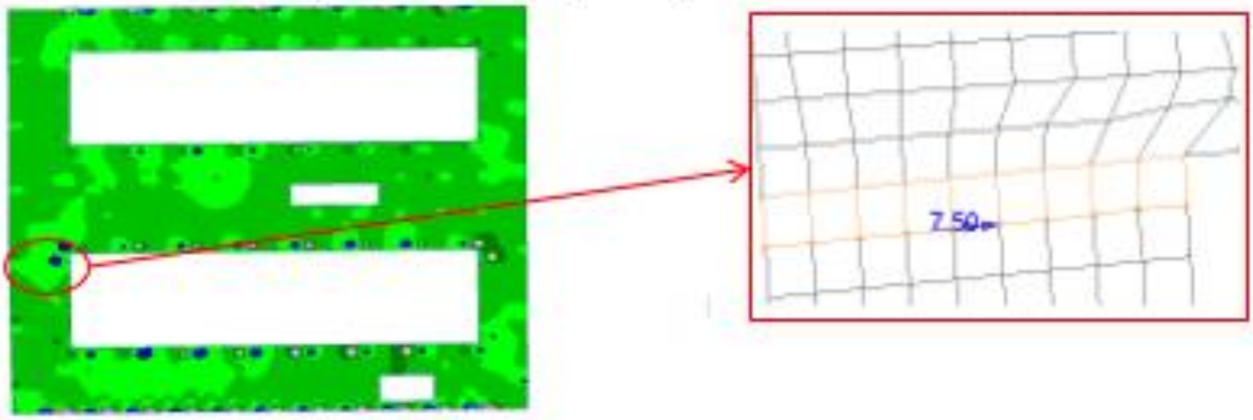
$f_{cc} = 23$ psi

$$MS = \left(\frac{F_{cc-core}}{f_{cc}}\right) - 1$$

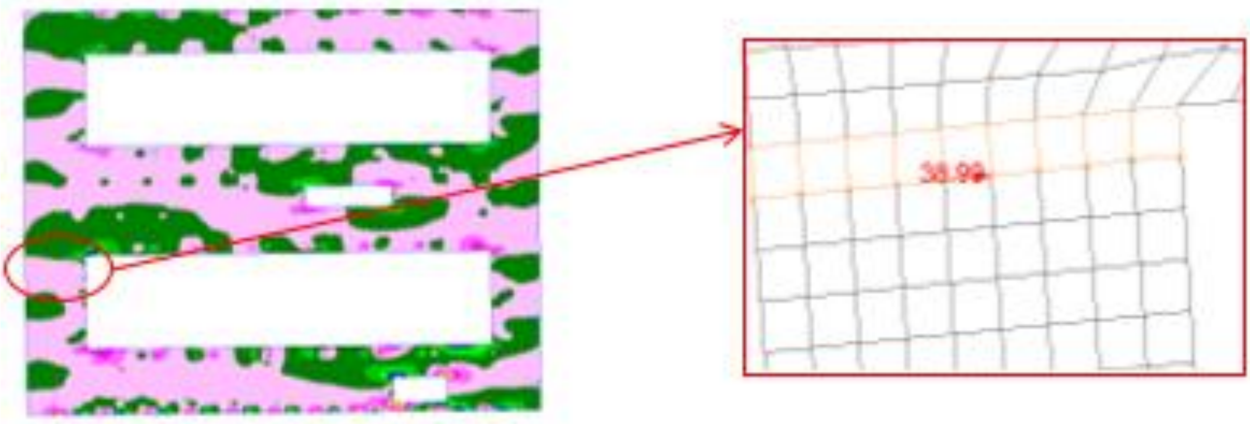
MS = Large Core Crushing



1G FWD, Moment Results, X Component



1G FWD, Forces Results, X Component



Inboard Panel

The Inboard Panel consist of two face shits (top and bottom) and a core between them (Figure 5.13).

Face Sheets: 2 ply (both sides) Phenolic Fiberglass Prepreg Fabric.

Core: 0.95 in thick, 1/8 in Cell Nomex Honeycomb.

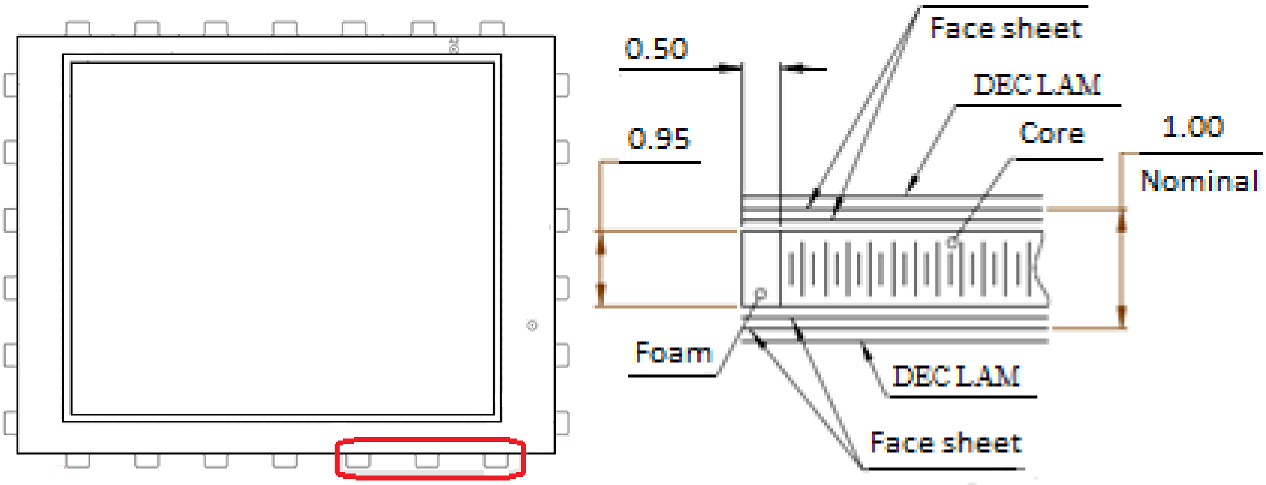


Figure 5.13. Initial Inboard Panel structure.

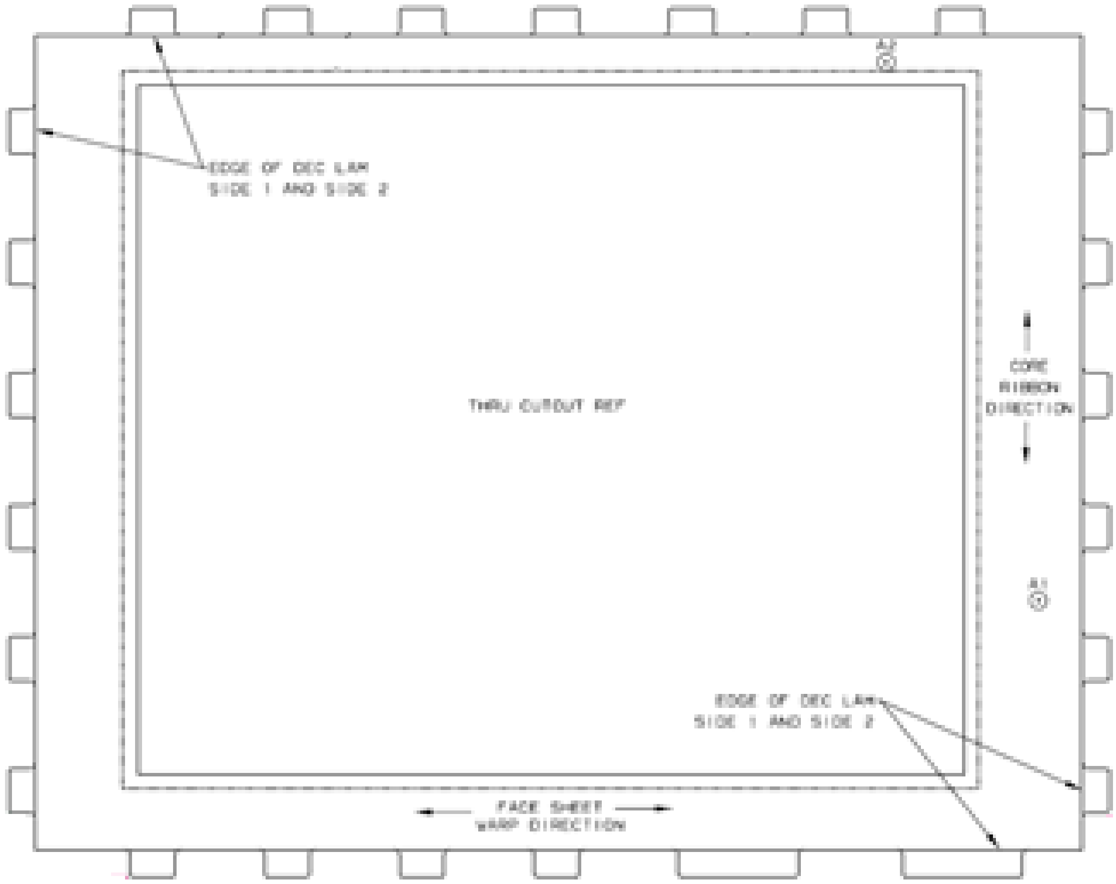


Figure 5.14. Updated Inboard Panel structure.

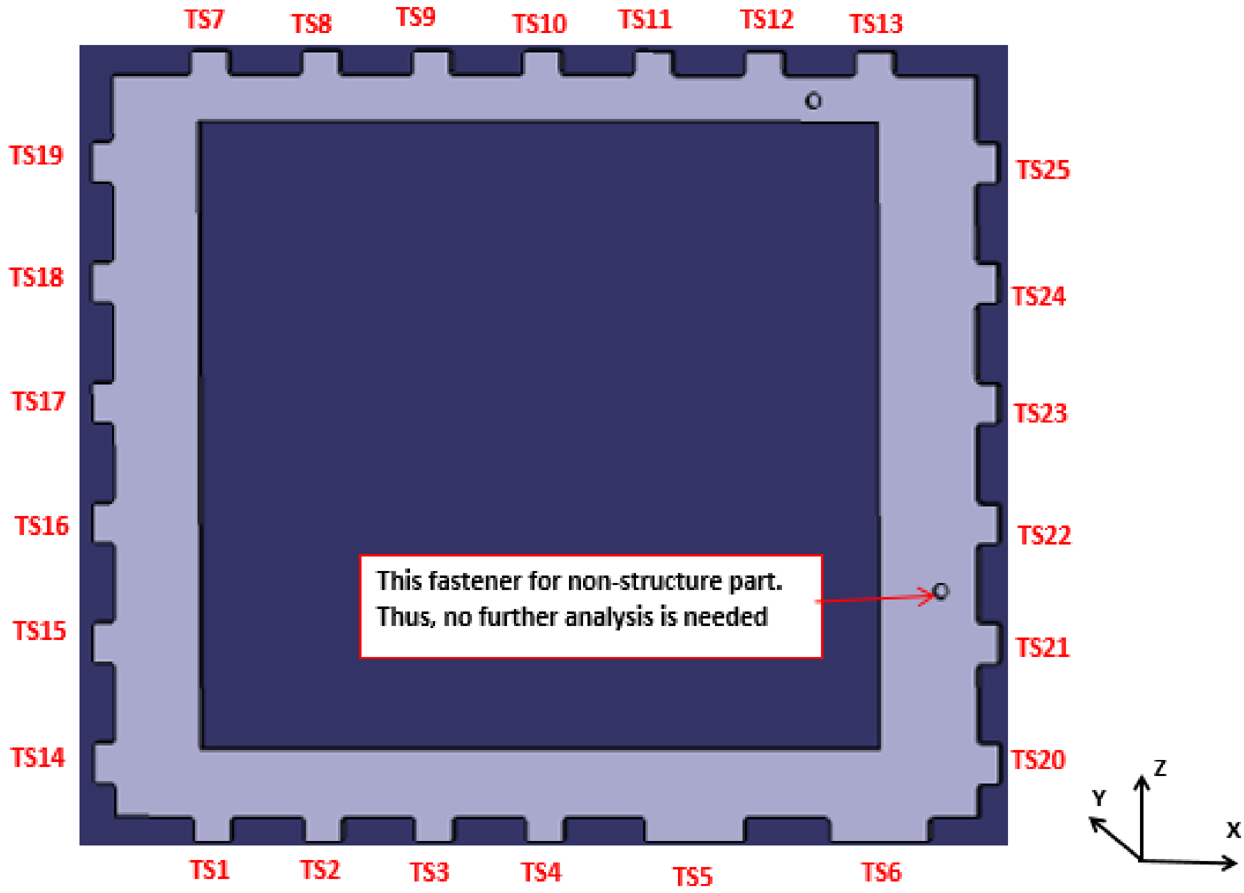


Figure 5.15. Numbering of Tabs/Slots and inserts of Inboard Panel.

The same method was used to analyze the Inboard Panel as for the panels discussed above.

Initially, the Inboard Panel had to be constructed as shown in Figure 5.13. However, during calculation process for three tabs (highlighted in red), negative MS values were obtained. To solve the problem, it was decided to make changes in the structure, namely, replace these 3 tabs with two others with larger area in order to increase the bonding area and, therefore, increase the allowable loads.

Due to this, the MS values increased to positive. The calculation for the updated version of the structure is presented below.

To get more accurate calculation result and to avoid material overruns, core ribbon direction and face sheet warp direction were taken into account.

Tab-Slot number	Shear Plane	Tension Direction
TS1-TS13	**X-Y**	**Z**
TS14-TS25	**Y-Z**	**X**

Table 5.21. Loads directions in Inboard Panel.

Tab/Slot	Tension Load	PerpShear Load	ParaShear Load	Bending Moment
	Pt [lb]	Ps [lb]	Psi [lb]	M [lb*in]
TS1	0	6	235	24
TS2	128	8	221	25
TS3	30	2	245	20
TS4	20	1	301	16
TS5	176	2	479	13
TS6	460	9	161	93
TS7	212	3	234	27
TS8	72	4	147	23
TS9	19	7	168	37
TS10	20	2	253	13
TS11	19	1	294	14
TS12	71	13	212	52
TS13	117	11	239	97
TS14	127	11	51	19
TS15	71	4	135	10
TS16	35	1	223	4
TS17	11	1	198	6
TS18	31	3	197	3
TS19	105	8	116	4
TS20	51	7	2	27
TS21	23	9	147	36
TS22	15	6	224	32
TS23	12	4	238	18
TS24	1	5	223	4
TS25	42	9	169	9

Table 5.22. Inboard Panel. Applied loads.

Tab/Slot	Tension Load	PerpShear Load	ParaShear Load	Bending Moment
	Pt_max	Ps_max	Psi_max	M_max
All	380	355	978	660

Table 5.23. Inboard Panel. Allowable loads.

Tab/Slot	Rt	Rs	Rsi	Rb
TS1	0.00	0.02	0.24	0.04
TS2	0.34	0.02	0.23	0.04
TS3	0.08	0.01	0.25	0.03
TS4	0.05	0.00	0.31	0.02
TS5	0.46	0.01	0.49	0.02
TS6	0.81	0.03	0.16	0.14
TS7	0.56	0.01	0.24	0.04
TS8	0.19	0.01	0.15	0.04
TS9	0.05	0.02	0.17	0.06
TS10	0.05	0.01	0.26	0.02
TS11	0.05	0.00	0.30	0.02
TS12	0.19	0.04	0.22	0.08

TS13	0.31	0.03	0.24	0.15
TS14	0.33	0.03	0.05	0.03
TS15	0.19	0.01	0.14	0.01
TS16	0.09	0.00	0.23	0.01
TS17	0.03	0.00	0.20	0.01
TS18	0.08	0.01	0.20	0.00
TS19	0.28	0.02	0.12	0.01
TS20	0.14	0.02	0.00	0.04
TS21	0.06	0.02	0.15	0.05
TS22	0.04	0.02	0.23	0.05
TS23	0.03	0.01	0.24	0.03
TS24	0.00	0.02	0.23	0.01
TS25	0.11	0.03	0.17	0.01

Table 5.24. Inboard Panel. Load ratio.

Tab/Slot	Tension & Shear	Bending & Tension	Bending & Shear
TS1	+3.17	Large	+1.04
TS2	+0.77	+1.03	+1.10
TS3	+2.04	+4.24	+0.99
TS4	+1.78	+5.87	+0.80
TS5	+0.05	+0.66	+0.43
TS6	**+0.03**	+0.11	+1.33
TS7	+0.25	+0.46	+1.04
TS8	+1.94	+1.96	+1.57
TS9	+3.51	+5.29	+1.39
TS10	+2.21	+5.83	+0.97
TS11	+1.84	+6.00	+0.82
TS12	+1.48	+1.84	+1.12
TS13	+0.81	+0.96	+0.94
TS14	+1.59	+1.06	+3.33
TS15	+2.09	+2.05	+1.69
TS16	+2.13	+3.90	+1.10
TS17	+3.35	+9.63	+1.22
TS18	+2.55	+4.36	+1.23
TS19	+1.54	+1.36	+1.91
TS20	+5.41	+2.65	+3.95
TS21	+3.76	+4.76	+1.55
TS22	+2.74	+6.42	+1.08
TS23	+2.64	+8.11	+1.03
TS24	+3.33	Large	+1.09
TS25	+2.52	+3.27	+1.41

Table 5.25. Inboard Panel. Margins of Safety.

All Margins of Safety of Tabs/Slots are positive, safety is provided.

Minimum MS is **+0.03** for TS6. Critical Load Case – 9G Forward. Mode of Failure is Ultimate Combined Load: Tension & Shear.

Additional check of Flexural Strength of Inboard Panel at sections A – A and B – B was carried out using Microsoft Excel.

By FEM the critical load case is 9G FWD

Face Sheets

Mi = Mmax / b
Mmax = 9*Mx (extracted from FEM, 1g FWD)
Mmax = 17,1 in-lb
b = 1,71 in
Mi = 10 in-lb/in
d = h-[(t1+t2)/2]
t1 = t2 = 0,025 in
h = 1,00 in (panel thickness)
d = 0,975 in
f1 = f2 = Mi/dt1
f1 = f2 = 410 psi
Fb = 14 ksi

$$MS = \left(\frac{F_b}{f_2}\right) - 1$$

Allowables 0.95" core:

Fult	14	ksi	face sheet tension/compression
Fcc	1358	psi	core compression
Fsu	345	psi	core Longitudinal shear

MS = **Large**

For Face Sheet bending

Core

V1 = Vmax / b
Vmax = 9*Vx (extracted from FEM, 1g FWD)
Vmax = 310,6 lb/in
V1 = 181,63 lb/in
c = Core Thickness
c = 0,950 in
fsc = V1/c
fsc = 191 psi
Fsw = 345 psi

$$MS = \left(\frac{F_{sw}}{f_{sc}}\right) - 1$$

MS = **+0,81** **For Core Shear**

$f_{cc} = M_{max}^2/b^2dD$

$D = E(I_i)$

$E_{cc} = 19000$ psi (core)

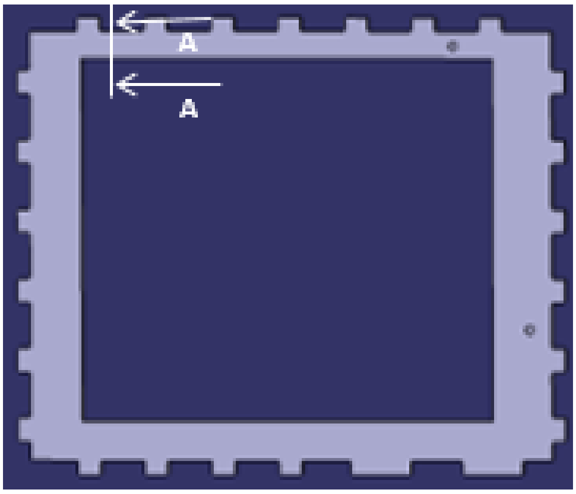
$I_i = d^2(t1t2/t1 + t2)$

$D = 226$ lb-in²/in

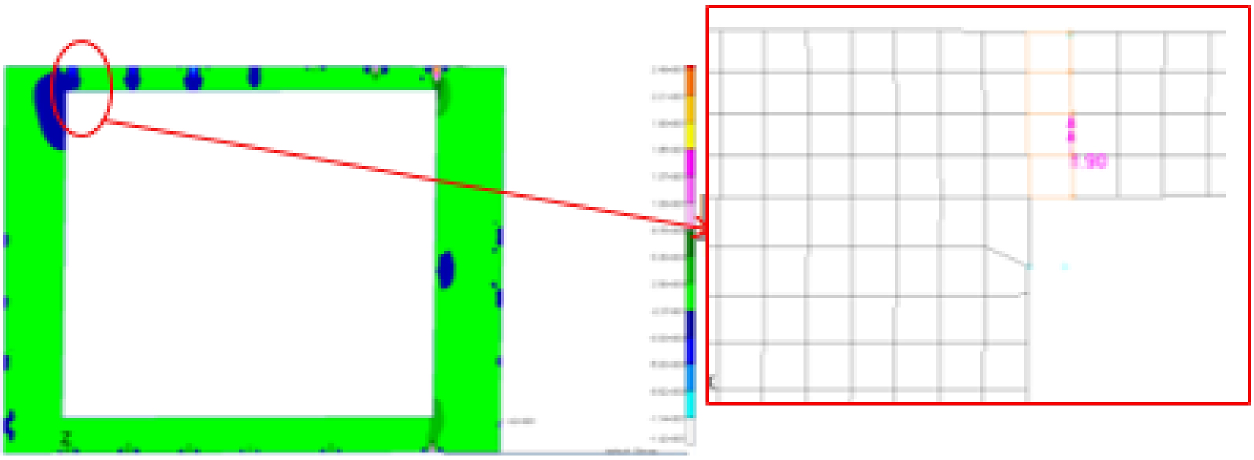
$f_{cc} = 0$ psi

$$MS = \left(\frac{F_{cc-core}}{f_{cc}}\right) - 1$$

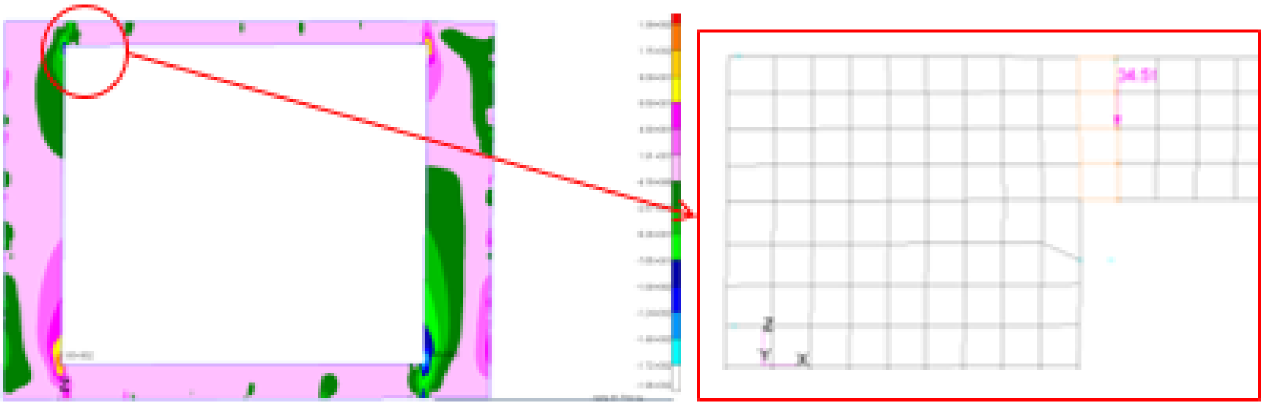
MS = Large Core Crushing



1G FWD, Moment Results, Z Component



1G FWD, Forces Results, Z Component



By FEM the critical load case is 9G FWD

Face Sheets

Mi = Mmax / b
Mmax = 9*Mx (extracted from FEM, 1g FWD)
Mmax = 9,3 in-lb
b = 1,71 in
Mi = 5,421 in-lb/in
d = h-[(t1+t2)/2]
t1 = t2 = 0,025 in
h = 1,00 in (panel thickness)
d = 0,975 in
f1 = f2 = Mi/dt1
f1 = f2 = 222 psi
Fb = 14 ksi

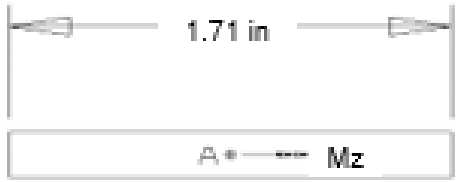
$$MS = \left(\frac{F_b}{f_2}\right) - 1$$

Allowables 0.95" core:

Fult	14	ksi	face sheet tension/compression
Fcc	1358	psi	core compression
Fsu	345	psi	core Longitudinal shear

MS = Large

For Face Sheet bending



B - B

Core

V1 = Vmax / b
Vmax = 9*Vx (extracted from FEM, 1g FWD)
Vmax = 156,0 lb/in
V1 = 91,21 lb/in
c = Core Thickness
c = 0,950 in
fsc = V1/c
fsc = 96 psi
Fsw = 345 psi

$$MS = \left(\frac{F_{sw}}{f_{sc}}\right) - 1$$

MS = +2,60 **For Core Shear**

$f_{cc} = M_{max}^2/b^2dD$

$D = E(I_i)$

$E_{cc} =$ 19000 psi (core)

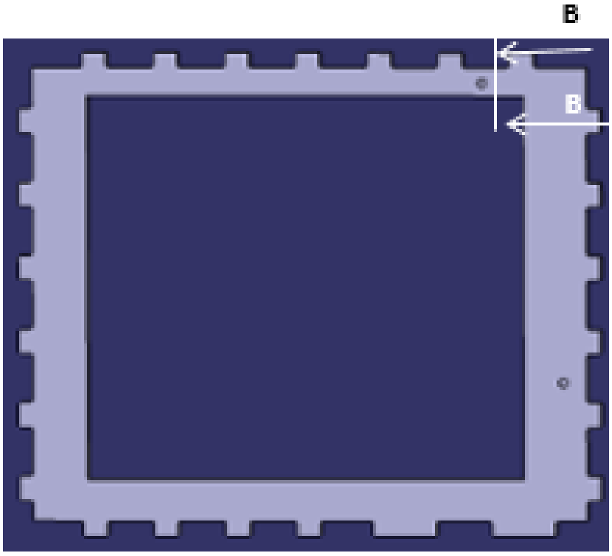
$I_i = d^2(t1t2/t1 + t2)$

$D =$ 226 lb-in^2/in

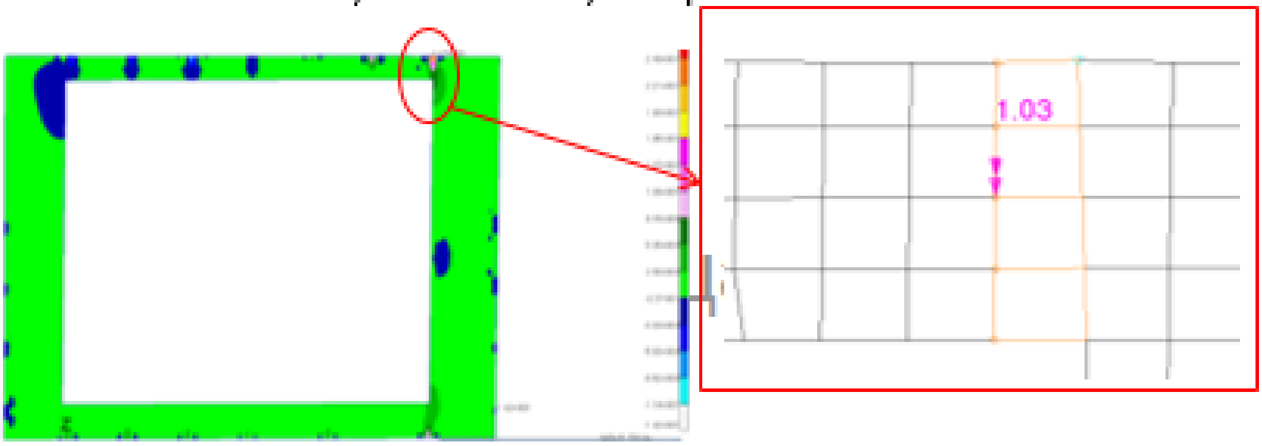
$f_{cc} =$ 0 psi

$$MS = \left(\frac{F_{cc-core}}{f_{cc}}\right) - 1$$

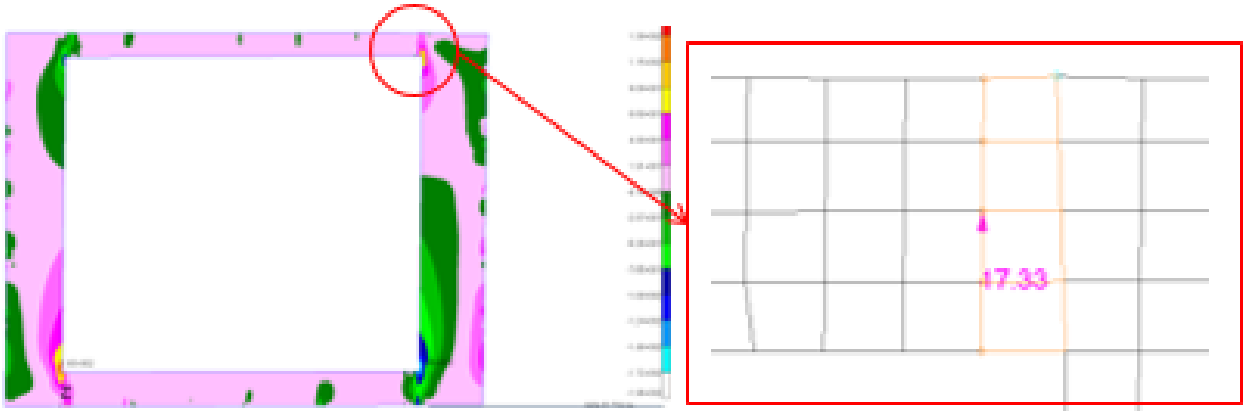
MS = Large Core Crushing



1G FWD, Moment Results, Z Component



1G FWD, Forces Results, Z Component



Work Surface Panel

The Work Surface Panel consist of two face shits (top and bottom) and a core between them (Figure 5.16).

Face Sheets: 2 ply (both sides) Phenolic Fiberglass Prepreg Fabric.

Core: 0.95 in thick, 1/8 in Cell Nomex Honeycomb.

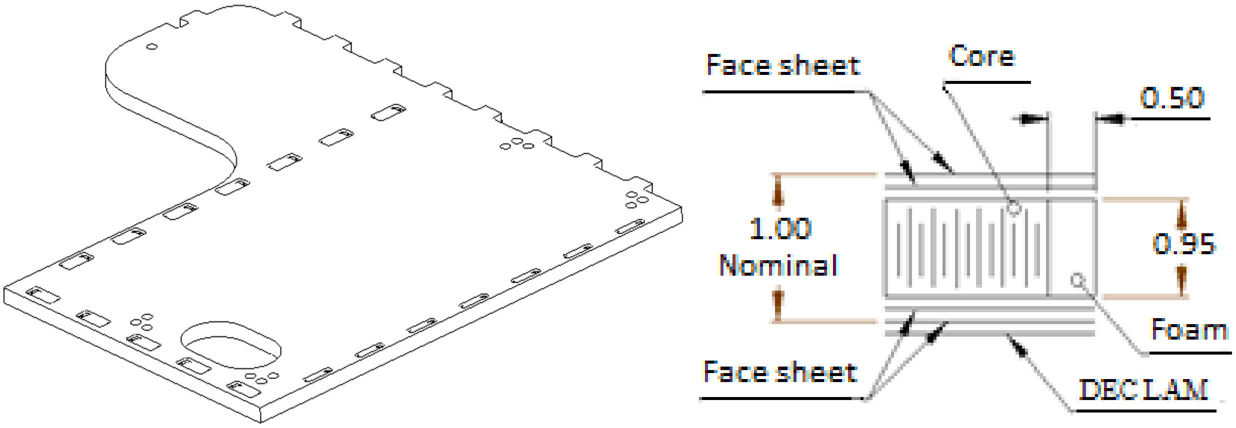


Figure 5.16. Work Surface Panel structure.

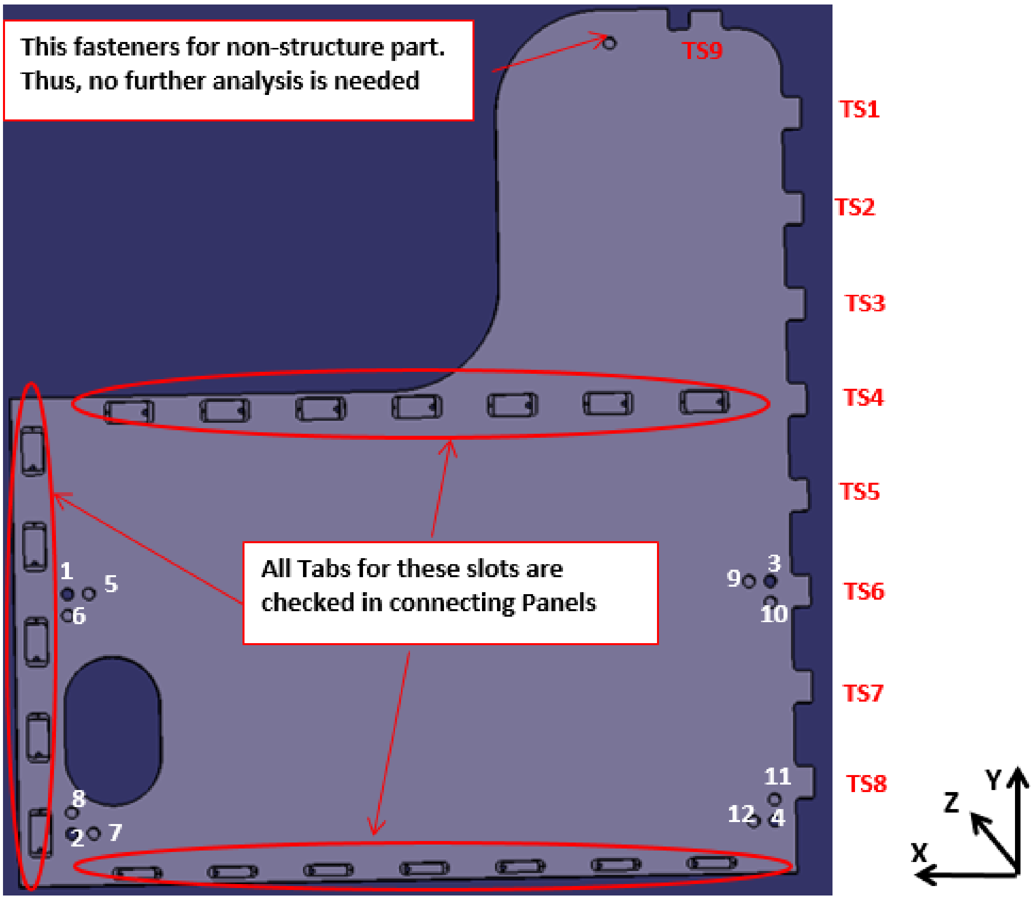


Figure 5.17. Numbering of Tabs/Slots and inserts of Work Surface Panel.

The same method was used to analyze the Work Surface Panel as for the panels discussed above.

Tab/Slot (or insert) number	Shear Plane	Tension Direction
TS1-TS8	**Y-Z**	**X**
TS9	**X-Z**	**Y**
1-12	**X-Y**	**Z**

Table 5.26. Loads directions in Work Surface Panel.

Tab/Slot	Tension Load Pt [lb]	PerpShear Load Ps [lb]	ParaShear Load Psi [lb]	Bending Moment M [lb*in]
TS1	22	46	18	63
TS2	36	8	33	90
TS3	56	32	33	76
TS4	67	48	9	76
TS5	47	12	26	110
TS6	22	114	4	157
TS7	36	114	41	151
TS8	46	154	34	75
TS9	46	28	122	70

Table 5.27. Work Surface Panel. Applied loads.

Tab/Slot	Tension Load Pt_max	PerpShear Load Ps_max	ParaShear Load Psi_max	Bending Moment M_max
All	380	355	978	660

Table 5.28. Work Surface Panel. Allowable loads.

Tab/Slot	Rt	Rs	Rsi	Rb
TS1	0.06	0.13	0.02	0.09
TS2	0.09	0.02	0.03	0.14
TS3	0.15	0.09	0.03	0.12
TS4	0.18	0.13	0.01	0.12
TS5	0.12	0.03	0.03	0.17
TS6	0.06	0.32	0.00	0.24
TS7	0.10	0.32	0.04	0.23
TS8	0.12	0.43	0.03	0.11
TS9	0.12	0.08	0.12	0.11

Table 5.29. Work Surface Panel. Load ratio.

Tab/Slot	Tension & Shear	Bending & Tension	Bending & Shear
TS1	+4.34	+4.06	+1.99
TS2	+6.87	+2.64	+1.70
TS3	+3.20	+2.05	+1.85
TS4	+2.22	+1.79	+1.74

TS5	+5.31	+2.01	+1.44
TS6	+1.65	+2.22	**+0.71**
TS7	+1.40	+1.88	+0.73
TS8	+0.80	+2.41	+0.82
TS9	+3.06	+2.44	+1.72

Table 5.30. Work Surface Panel. Margins of Safety.

All Margins of Safety of Tabs/Slots are positive, safety is provided.

Minimum MS is **+0.71** for TS6. Critical Load Case – 9G Forward. Mode of Failure is Ultimate Combined Load: Bending & Shear.

At Work Surface Panel vertical loads carried by nut plates (Figure 5.18) and not critical for inserts. Critical mode of failure is bearing.

$$MS_{bearing} = V_{all\_bearing} / V_{app\_bearing} - 1$$

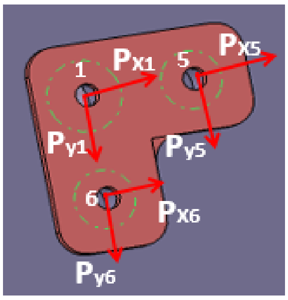


Figure 5.18. Reinforcing plate.

Insert	Applied Load			Edge distance [in]	Allowable Load [lb]		MS
	Papp [lb]		Vapp [lb]		Pmax	Vmax	
1	Vertical loads carried by nut plates and not critical for inserts	0	51	0.5	318	245	**+3.79**
2		0	31	0.5	318	245	**+6.87**
3		0	61	0.5	318	245	**+3.03**
4		0	48	0.5	318	245	**+4.13**
5		0	135	0.5	172	200	**+0.48**
6		0	135	0.5	172	200	**+0.48**
7		0	82	0.5	172	200	**+1.43**
8		0	82	0.5	172	200	**+1.43**
9		0	161	0.5	172	200	**+0.24**
10		0	161	0.5	172	200	**+0.24**
11		0	126	0.5	172	200	**+0.58**
12		0	126	0.5	172	200	**+0.58**

Table 5.31. Work Surface Panel. Insert data.

All insert's Margins of Safety are positive, safety is provided.

Minimum MS is **+0.24** for insert №10. Critical Load Case – 9G Forward. Mode of Failure is Ultimate Bearing.

Lower Forward Panel

The Lower Forward Panel consist of two face shits (top and bottom) and a core between them (Figure 5.19).

Face Sheets: 2 ply (both sides) Phenolic Fiberglass Prepreg Fabric.

Core: 0.95 in thick, 1/8 in (3/16 O.X. in) Cell Nomex Honeycomb.

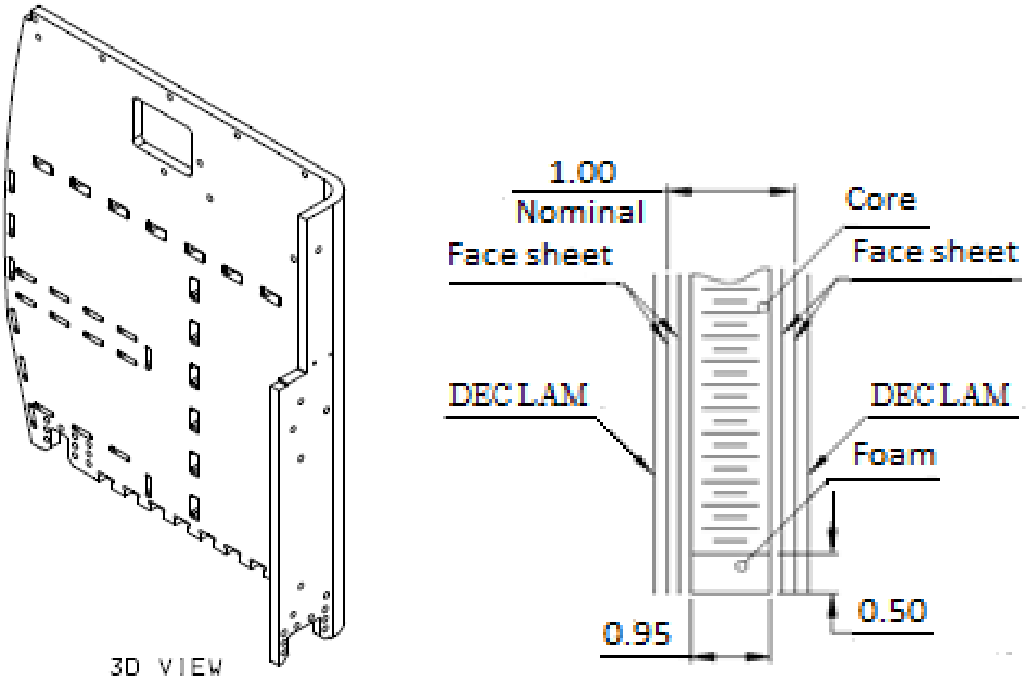


Figure 5.19. Lower Forward Panel structure.

To analyze the Lower Forward Panel, the same method was used as for the panels discussed above.

Since initially negative values were obtained for several Margins of Safety, it was decided to increase the core density. The calculations below are presented with this in mind.

Tab/Slot (or insert) number	Shear Plane	Tension Direction
TS1-TS8	**X-Y**	**Z**
1-10, 19-23 & DB1	**Y-Z**	**X**
11-18, 24-27	**X-Z**	**Y**

Table 5.32. Loads directions in Lower Forward Panel.

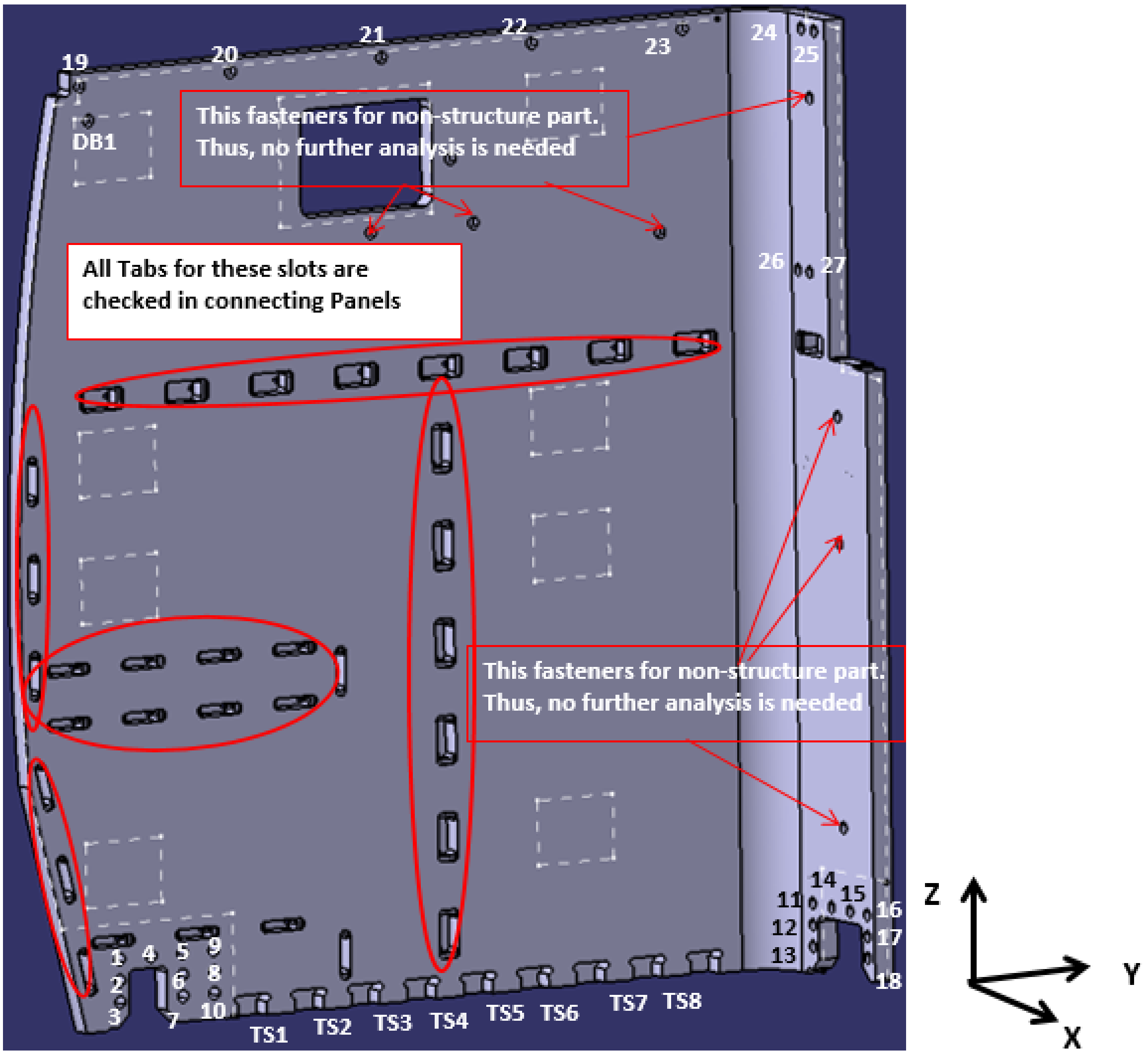


Figure 5.20. Numbering of Tabs/Slots and inserts of Lower Forward Panel.

Tab/Slot	Tension Load	PerpShear Load	ParaShear Load	Bending Moment
	Pt [lb]	Ps [lb]	Psi [lb]	M [lb*in]
TS1	1	26	59	57
TS2	8	60	12	62
TS3	24	53	11	58
TS4	38	53	19	52
TS5	12	14	53	59
TS6	9	6	61	67
TS7	19	2	79	67
TS8	69	16	111	53

Table 5.33. Lower Forward Panel. Applied loads.

Tab/Slot	Tension Load	PerpShear Load	ParaShear Load	Bending Moment
	Pt_max	Ps_max	Psi_max	M_max
All	299	299	1004	480

Table 5.34. Lower Forward Panel. Allowable loads.

Tab/Slot	Rt	Rs	Rsi	Rb
TS1	0.00	0.09	0.06	0.12
TS2	0.03	0.20	0.01	0.13
TS3	0.08	0.18	0.01	0.12
TS4	0.13	0.18	0.02	0.11
TS5	0.04	0.05	0.05	0.12
TS6	0.03	0.02	0.06	0.14
TS7	0.06	0.01	0.08	0.14
TS8	0.23	0.05	0.11	0.11

Table 5.35. Lower Forward Panel. Load ratio.

Tab/Slot	Tension & Shear	Bending & Tension	Bending & Shear
TS1	+9.97	+6.94	+1.81
TS2	+3.40	+4.67	+1.44
TS3	+2.86	+3.05	+1.55
TS4	+2.26	+2.34	+1.67
TS5	+9.83	+4.24	+1.82
TS6	Large	+4.30	+1.67
TS7	+6.08	+3.24	+1.68
TS8	+1.91	**+1.40**	+1.85

Table 5.36. Lower Forward Panel. Margins of Safety.

All Margins of Safety of Tabs/Slots are positive, safety is provided.

Minimum MS is **+1.40** for TS8. Critical Load Case – 9G Forward. Mode of Failure is Ultimate Combined Load: Bending & Tension.

Insert	Applied Load		Edge distance [in]	Allowable Load [lb]		Rt	Rs	MS
	Papp [lb]	Vapp [lb]		Pmax	Vmax	Papp/Pmax	Vapp/Vmax	
1	2	620	0.5	449	385	0.004	1.609	**-0.38**
2	11	346	0.5	449	385	0.026	0.899	**+0.08**
3	23	379	0.5	449	385	0.051	0.985	**-0.03**
4	6	429	0.5	449	385	0.014	1.115	**-0.11**
5	3	282	0.5	449	385	0.007	0.731	**+0.35**
6	7	131	0.5	449	385	0.016	0.341	**+1.80**
7	27	102	0.5	449	385	0.060	0.265	**+2.08**
8	3	256	1.5	449	385	0.006	0.664	**+0.49**
9	0	134	1.5	449	385	0.000	0.349	**+1.87**
10	32	100	0.5	449	385	0.070	0.260	**+2.03**
11	5	230	0.5	449	385	0.012	0.597	**+0.64**
12	0	131	0.5	449	385	0.001	0.340	**+1.94**
13	9	180	0.5	449	385	0.021	0.467	**+1.05**
14	4	105	0.5	449	385	0.009	0.272	**+2.55**

15	2	101	0.5	449	385	0.004	0.262	**+2.76**
16	3	146	0.5	449	385	0.008	0.380	**+1.58**
17	2	53	0.5	449	385	0.005	0.138	**+5.97**
18	1	33	0.5	449	385	0.003	0.085	**Large**
19	85	213	0.5	328	325	0.260	0.656	**+0.09**
20	14	44	0.5	238	200	0.061	0.222	**+2.53**
21	1	32	0.5	238	200	0.003	0.158	**+5.21**
22	8	43	0.5	238	200	0.033	0.216	**+3.00**
23	14	54	0.5	238	200	0.060	0.268	**+2.05**
24	59	14	0.5	180	143	0.329	0.101	**+1.33**
25	11	130	0.5	180	143	0.059	0.906	**+0.04**
26	15	39	0.5	180	143	0.086	0.275	**+1.77**
27	42	38	0.5	180	143	0.235	0.266	**+1.00**
DB1	64	1	1.5	133	377	0.478	0.004	**+1.08**
Total Shear Load (1-10)		2779						

Table 5.37. Lower Forward Panel. Insert data.

Negative MS values were obtained for inserts №1, 3 & 4. To increase them, the Adhesive Bond was used for connection Fitting and Panel (inserts 1-10).

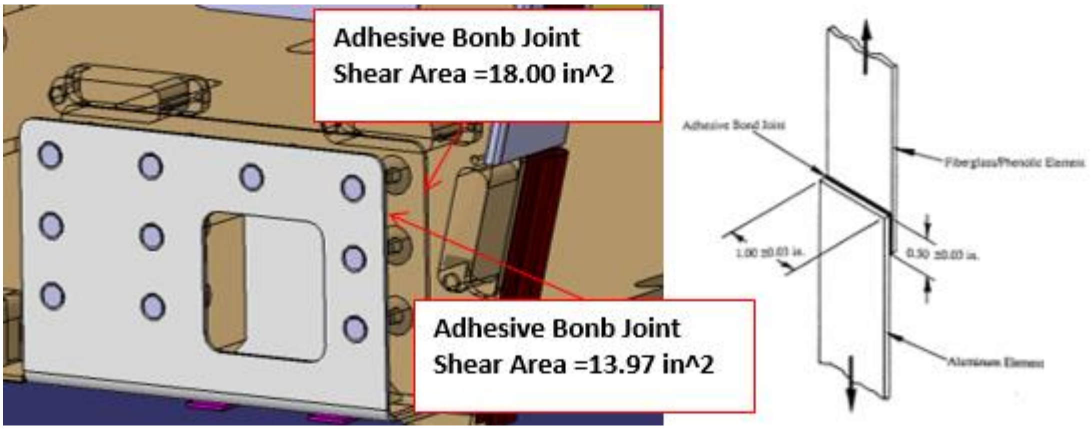


Figure 5.21. Adhesive Bond for connecting Fitting with Panel.

Additional calculation was performed for Adhesive Bond:

- applied load: P_{app} = Total Shear Load / 2 = 2779/2 = 1389.5 lb;
- allowable load (according to the BOEING regulatory document, P_{all} per 1 square inch equals 1020 psi): P_{all} = 1020*13.97 = 14249.4 lb;
- Margin of Safety: MS = P_{all} / $P_{app} - 1$ = 14249.4/1389.5 – 1= **+9.25.**

Inserts №1, 3 & 4 are critical. Critical Load Case – 9G Forward. Mode of Failure is Ultimate Combined Load: Tension & Shear. For these inserts, Margins of Safety is negative, but since Adhesive Bond was added at critical place and Margin of Safety of Adhesive Bond is positive (**+9.25**), the calculation is considered acceptable and safety is ensured.

Upper AFT Panel

The Upper AFT Panel consist of two face shits (top and bottom) and a core between them (Figure 5.22).

Face Sheets: 2 ply (both sides) Phenolic Fiberglass Prepreg Fabric.

Core: 0.47 in thick, 1/8 in Cell Nomex Honeycomb.

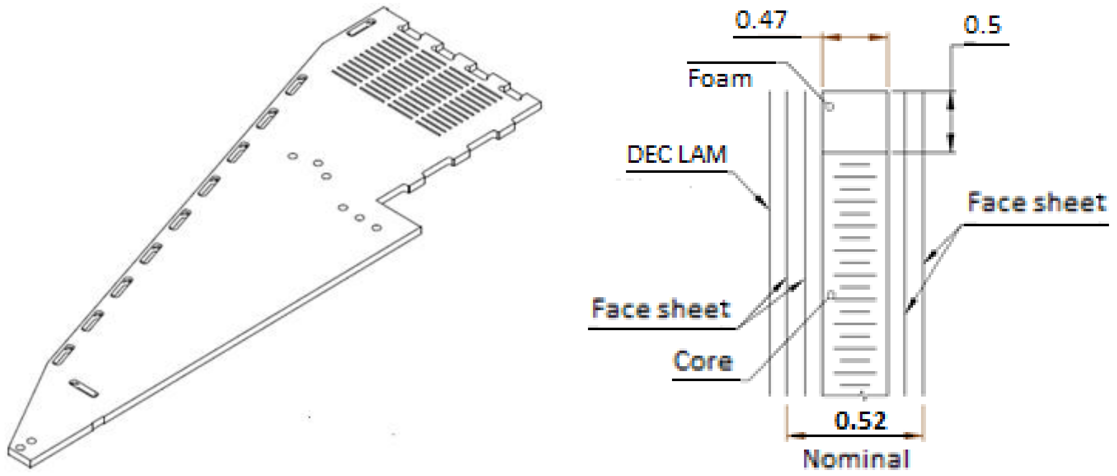


Figure 5.22. Upper AFT Panel structure.

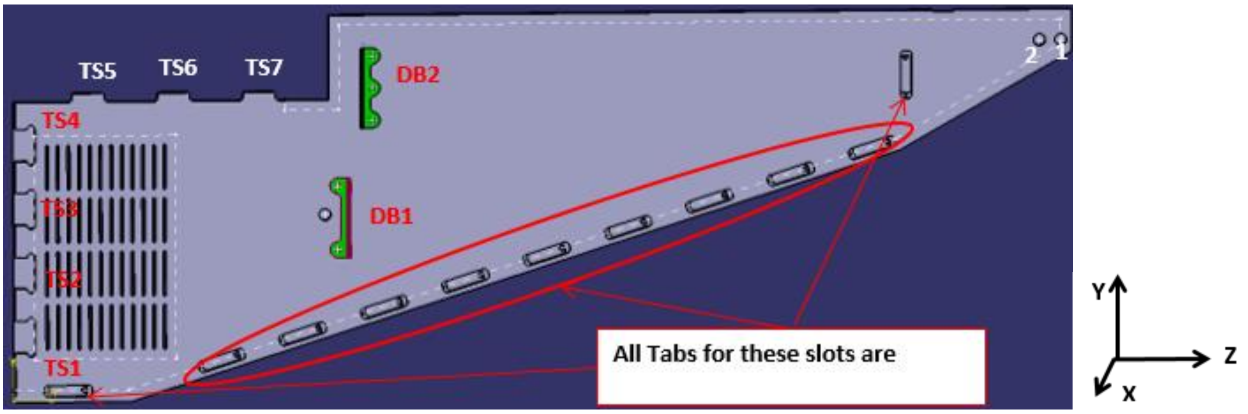


Figure 5.23. Numbering of Tabs/Slots and inserts of Upper AFT Panel.

To analyze the Upper AFT Panel, the same method was used as for the panels discussed above.

Tab/Slot (or insert) number	Shear Plane	Tension Direction
TS1-TS4	**X-Y**	**Z**
TS5-TS7	**X-Z**	**Y**
1-12 & DB1-DB2	**Y-Z**	**X**

Table 5.38. Loads directions in Upper AFT Panel.

Tab/Slot	Tension Load	PerpShear Load	ParaShear Load	Bending Moment
	Pt [lb]	Ps [lb]	Psi [lb]	M [lb*in]
TS1	54	1	26	30
TS2	27	2	27	27
TS3	52	1	24	29
TS4	79	0	19	24
TS5	6	4	48	1
TS6	1	7	37	4
TS7	50	29	34	19

Table 5.39. Upper AFT Panel. Applied loads.

Tab/Slot	Tension Load	PerpShear Load	ParaShear Load	Bending Moment
	Pt_max	Ps_max	Psi_max	M_max
TS1-TS4	369	247	892	246
TS5-TS7	488	322	804	222

Table 5.40. Upper AFT Panel. Allowable loads.

Tab/Slot	Rt	Rs	Rsi	Rb
TS1	0.15	0.00	0.03	0.12
TS2	0.07	0.01	0.03	0.11
TS3	0.14	0.00	0.03	0.12
TS4	0.21	0.00	0.02	0.10
TS5	0.01	0.01	0.06	0.01
TS6	0.00	0.02	0.05	0.02
TS7	0.10	0.09	0.04	0.09

Table 5.41. Upper AFT Panel. Load ratio.

Tab/Slot	Tension & Shear	Bending & Tension	Bending & Shear
TS1	+4.69	+2.04	+1.88
TS2	+8.60	+3.37	+2.02
TS3	+5.00	+2.13	+1.90
TS4	+3.27	**+1.56**	+2.20
TS5	Large	Large	+3.09
TS6	Large	Large	+3.67
TS7	+4.19	+2.94	+2.27

Table 5.42. Upper AFT Panel. Margins of Safety.

All Margins of Safety of Tabs/Slots are positive, safety is provided.

Minimum MS is **+1.56** for TS4. Critical Load Case – 9G Forward. Mode of Failure is Ultimate Combined Load: Bending & Tension.

Insert	Applied Load		Edge distance [in]	Allowable Load [lb]		Rt	Rs	MS
	Papp [lb]	Vapp [lb]		Pmax	Vmax	Papp/Pmax	Vapp/Vmax	
1	14	10	0.5	118	310	0.120	0.031	**+5.62**
2	18	13	0.5	172	200	0.103	0.067	**+4.87**
DB1	30	22	1.5	139	704	0.216	0.032	**+3.03**
DB2	18	64	1.5	209	986	0.085	0.065	**+5.66**

Table 5.43. Upper AFT Panel. Insert data.

All insert's and Dog-Bone's Margins of Safety are positive, safety is provided.

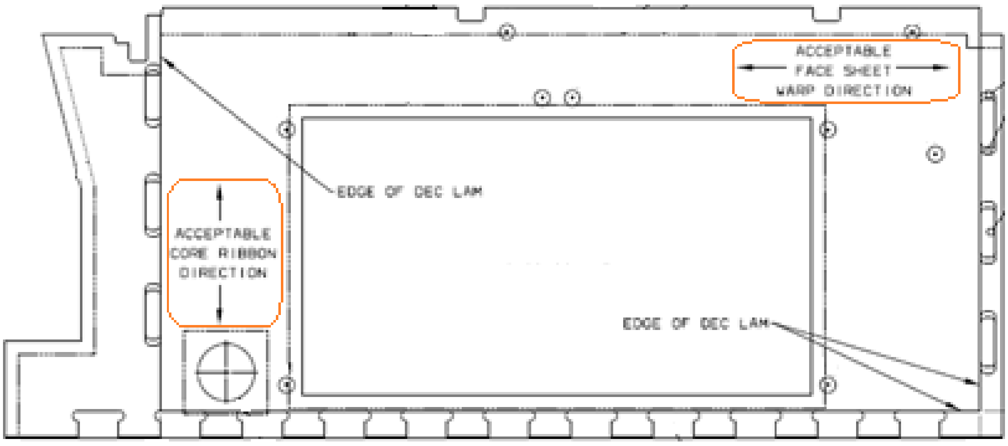
Minimum MS is **+3.03** for DB1. Critical Load Case – 3G Right. Mode of Failure is Ultimate Combined Load: Shear & Tension.

Upper Cab INBD Panel

The Upper Cab INBD Panel consist of two face shits (top and bottom) and a core between them (Figure 5.24).

Face Sheets: 2 ply (both sides) Phenolic Fiberglass Prepreg Fabric.

Core: 0.47 in thick, 1/8 in Cell Nomex Honeycomb.



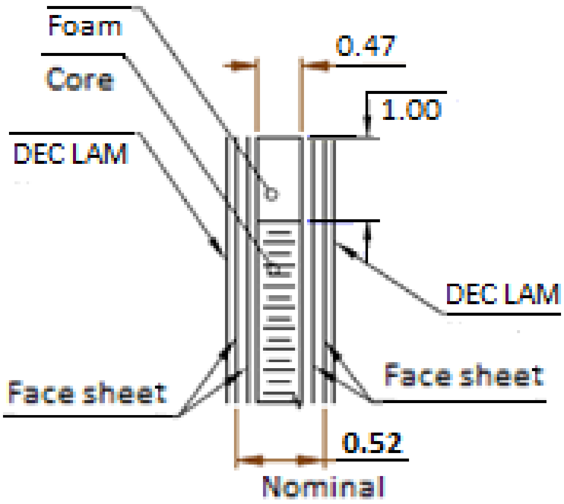


Figure 5.24. Upper Cab INBD Panel structure.

To analyze the Upper Cab INBD Panel, the same method was used as for the panels discussed above. To get more accurate calculation result and to avoid material overruns, core ribbon direction and face sheet warp direction were taken into account.

Tab/Slot (or insert) number	Shear Plane	Tension Direction
TS1-TS11	**X-Y**	**Z**
1-8 & DB1-DB3	**X-Z**	**Y**

Table 5.44. Loads directions in Upper Cab INBD Panel.

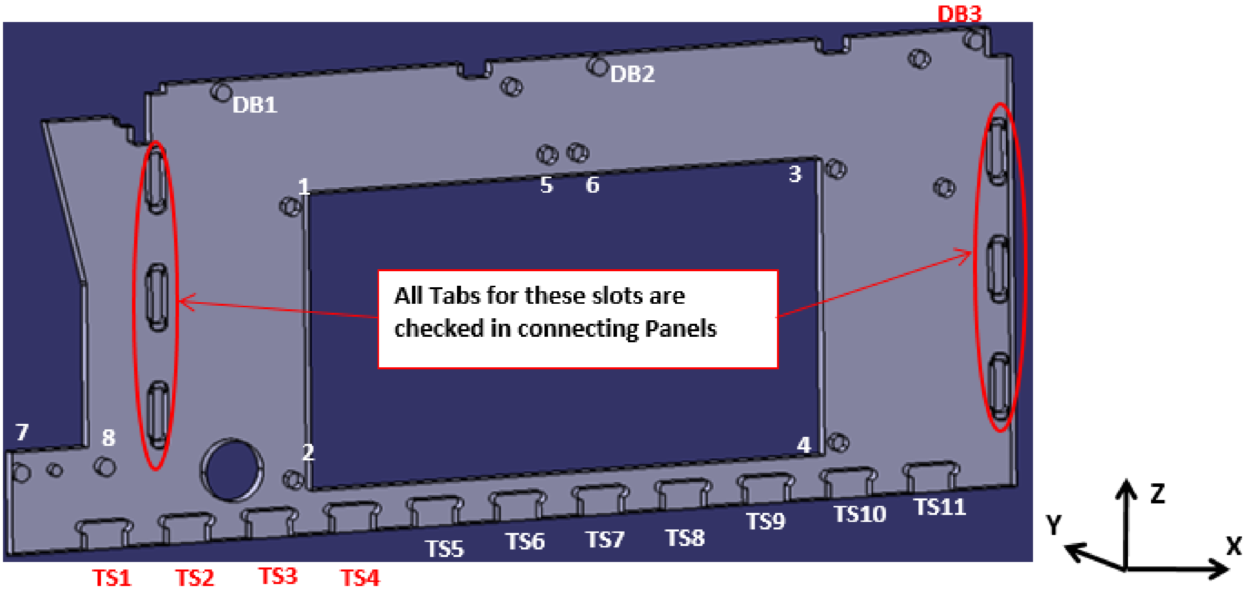


Figure 5.25. Numbering of Tabs/Slots and inserts of Upper Cab INBD Panel.

Tab/Slot	Tension Load	PerpShear Load	ParaShear Load	Bending Moment
	Pt [lb]	Ps [lb]	Psi [lb]	M [lb*in]
TS1	174	34	56	30
TS2	5	3	78	2
TS3	48	20	58	17
TS4	56	11	114	22
TS5	13	4	55	4
TS6	14	1	27	1
TS7	14	1	32	1
TS8	4	1	54	3
TS9	14	8	59	6
TS10	72	22	37	16
TS11	109	9	54	30

Table 5.45. Upper Cab INBD Panel. Applied loads.

Tab/Slot	Tension Load	PerpShear Load	ParaShear Load	Bending Moment
	Pt_max	Ps_max	Psi_max	M_max
TS1-TS4	369	247	892	246

Table 5.46. Upper Cab INBD Panel. Allowable loads.

Tab/Slot	Rt	Rs	Rsi	Rb
TS1	0.47	0.14	0.06	0.12
TS2	0.01	0.01	0.09	0.01
TS3	0.13	0.08	0.07	0.07
TS4	0.15	0.04	0.13	0.09
TS5	0.04	0.02	0.06	0.01
TS6	0.04	0.00	0.03	0.00

TS7	0.04	0.00	0.04	0.00
TS8	0.01	0.01	0.06	0.01
TS9	0.04	0.03	0.07	0.02
TS10	0.20	0.09	0.04	0.07
TS11	0.30	0.04	0.06	0.12

Table 5.47. Upper Cab INBD Panel. Load ratio.

Tab/Slot	Tension & Shear	Bending & Tension	Bending & Shear
TS1	+0.64	**+0.56**	+1.66
TS2	+8.92	Large	+2.39
TS3	+3.69	+2.58	+2.67
TS4	+2.57	+2.14	+1.72
TS5	+9.26	+7.84	+3.04
TS6	Large	+7.91	+4.71
TS7	Large	+7.74	+4.26
TS8	Large	Large	+3.05
TS9	+8.66	+7.42	+2.86
TS10	+2.48	+1.80	+2.68
TS11	+1.81	+1.06	+1.86

Table 5.48. Upper Cab INBD Panel. Margins of Safety.

All Margins of Safety of Tabs/Slots are positive, safety is provided.

Minimum MS is **+0.56** for TS1. Critical Load Case – 9G Forward. Mode of Failure is Ultimate Combined Load: Bending & Tension.

Insert	Applied Load		Edge distance	Allowable Load [lb]		Rt	Rs	MS
	Papp [lb]	Vapp [lb]	[in]	Pmax	Vmax	Papp/Pmax	Vapp/Vmax	
1	43	135	0.5	118	310	0.361	0.434	**+0.26**
2	45	147	0.5	118	310	0.382	0.474	**+0.17**
3	42	142	0.5	118	310	0.359	0.458	**+0.22**
4	45	161	0.5	118	310	0.379	0.518	**+0.11**
5	17	0	0.5	172	200	0.097	0.000	**+9.34**
6	17	0	0.5	172	200	0.097	0.000	**+9.34**
7	18	54	0.41	172	200	0.104	0.271	**+1.67**
8	33	223	1.5	172	375	0.193	0.595	**+0.27**
DB1	4	163	0.43	105	256	0.038	0.639	**+0.48**
DB2	4	201	0.43	105	256	0.038	0.786	**+0.21**
DB3	17	107	0.43	105	256	0.165	0.417	**+0.72**

Table 5.49. Upper Cab INBD Panel. Insert data.

All insert's and Dog-Bone’s Margins of Safety are positive, safety is provided.

Minimum MS is **+0.11** for insert №4. Critical Load Case – 9G Forward. Mode of Failure is Ultimate Combined Load: Shear & Tension.

6. Conclusions

In the process of analysing, Margins of Safety were determined for joints of panels of flight attendant Personal Work Station structure. For all problem areas, an option was developed to improve the structural carrying capacity. Ultimately, the calculation results showed that all Margins of Safety are positive, which means that the safety conditions are met.

Therefore, we can conclude that the structure of flight attendant Personal Work Station, even if it is not attached to the overhead aircraft structure, satisfies the strength conditions taking into account significant overloads that may occur during an emergency landing.

The solution of this problem allowed us to formulate an important conclusion for practical use that the considered structure of the flight attendant Personal Work Station can be installed in an aircraft by attaching it only to passenger floor of an aircraft.

The developed finite element model can be used to calculate the strength of other similar structures.

Bibliography

1. User's Guide "MSC Nastran 2018.2 Documentation"
2. УЧЕБНОЕ ПОСОБИЕ "MSC.PATRAN В ИНЖЕНЕРНЫХ ЗАДАЧАХ", Яхно Б.О., Гладский М.Н.
3. Lecture Notes by R. B. Agarwal
4. https://en.wikipedia.org/wiki/Finite_element_method
5. https://patents.google.com/patent/EP2040980A2
6. Airworthiness Standards FAA FAR Part 25