**НАЦІОНАЛЬНИЙ ТЕХНІЧНИЙ УНІВЕРСИТЕТ УКРАЇНИ
«КИЇВСЬКИЙ ПОЛІТЕХНІЧНИЙ ІНСТИТУТ
імені ІГОРЯ СІКОРСЬКОГО»**

МЕХАНІКО-МАШИНОБУДІВНИЙ ІНСТИТУТ

КАФЕДРА ДИНАМІКИ І МІЦНОСТІ МАШИН ТА ОПОРУ МАТЕРІАЛІВ

«На правах рукопису»
УДК ________________

«До захисту допущено»

Завідувач кафедри

______ Сергій ПИСКУНОВ

«___»____________2020 р.

Магістерська дисертація

на здобуття ступеня магістра

за освітньо-професійною програмою «Динаміка і міцність машин»

зі спеціальності 131 «Прикладна механіка»

на тему: «Визначення напружено-деформованого стану конструкції центра контролю пасажирського відеообладнання літака»

Виконав (-ла):
студент (-ка) VI курсу, групи мп-91мп
Малаховська Ганна Сергіївна ___________

Керівник:
д. т. н., професор
Бабенко Андрій Єлисейович ___________

Рецензент:
д.т.н., проф.
Данильченко Ю.М. ___________

Засвідчую, що у цій магістерській дисертації немає запозичень з праць інших авторів без відповідних посилань.
Студент (-ка)

Київ – 2020 року

Національний технічний університет України
«Київський політехнічний інститут імені Ігоря Сікорського»
Механіко-машинобудівний інститут
Кафедра динаміки і міцності машин та опору матеріалів

Рівень вищої освіти – другий (магістерський)

Спеціальність – 131 «Прикладна механіка»

Освітньо-професійна програма – «Динаміка і міцність машин»

ЗАТВЕРДЖУЮ

Завідувач кафедри

________ Сергій ПИСКУНОВ

«___»____________2020 р.

ЗАВДАННЯ
на магістерську дисертацію студенту

Малаховська Ганна Сергіївна

1. Тема дисертації «Визначення напружено-деформованого стану конструкції центра контролю пасажирського відеообладнання літака», науковий керівник дисертації Бабенко Андрій Єлисейович, д. т. н., професор , затверджені наказом по університету від «   »   2020 р. № ____

2. Термін подання студентом дисертації: 10.12.2020 р.

3. Об'єкт дослідження: композитна конструкція центру контролю пасажирського відеообладнання літака

4. Предмет дослідження: використання скінчено-елементного методу для визначення напружено-деформованого стану композитної сендвіч конструкції та отримання задовільних міцностних характеристик конструкції літака.

5. Перелік завдань, які потрібно розробити:

1) Огляд особливостей використання скінчено-елементного методу для композитної конструкції літака та базових допущень.
2) Визначення методів дослідження та аналізу міцності композитної панелі, критеріїв руйнування та основних переваг і допущень визначеного методу при аналізі міцності композитної конструкції.
3) Фізичне моделювання композитної конструкції за допомогою методу скінченних елементів та програмних комплексів MSC Patran та MSC

Nastran, використовуючи метод безпечного руйнування для оптимізації та отримання задовільних експлуатаційних характеристик.

4) Аналітичні розрахунки композитних з’єднань на достатню міцність.
5) Розробка стартап–проекту.

6. Орієнтовний перелік графічного (ілюстративного) матеріалу – 6 і більше.

7. Орієнтовний перелік публікацій – 1 і більше.

8. Дата видачі завдання: 01.09.2020 р.

Календарний план

№ з/п	Назва етапів виконання магістерської дисертації	Термін виконання етапів магістерської дисертації	Примітка
1	Аналіз літератури	01.09.20 - 19.09.20	
2	Огляд та визначення методу аналізу міцності композитної конструкції	20.09.20 – 09.10.20	
3	Фізичне моделювання конструкції	10.10.20 – 02.11.20	
4	Аналітичні розрахунки з’єднань	03.11.20 – 19.11.20	
5	Стартап-проект та висновки	20.11.20 – 10.12.20	

Студент Ганна МАЛАХОВСЬКА

Науковий керівник _____________ Андрій БАБЕНКО

CONTENTS

ABSTRACT

The master`s degree dissertation for the amount of work is 74 pages, 55 figures, 23 tables, and contains 12 literature.

The object of the work is the Cabin Entertainment Center (CEC) of passenger aircraft.

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

Relevance: Engineering composite structures that have complex geometry and loads are very difficult to analyze or have no theoretical solution. The relevance is to solve this problem using the finite element method and software packages MSC Patran and MSC Nastran.

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

As a result of this work, it was proved that the CEC withstands all specified overloads without damage and safety requirements met.

РЕФЕРАТ

Дана магістерська дисертація за обсягом роботи складає 7 сторінок, 55 ілюстрацію, 23 таблицю та містить 12 літературних джерел.

Об’єктом дослідження є центр контролю відеобладнання пасажирського літака.

Головна ціль даної дисертації – детальний аналіз центра контролю відеобладнання пасажирського літака згідно з вимогами безпеки.

Актуальність: Інженерні композитні споруди, що мають складну геометрію та навантаження дуже важкі для аналізу або не мають теоретичного рішення. Актуальність полягає у вирішенні цієї проблеми за допомогою методу скінченних елементів та програмних комплексів MSC Patran та MSC Nastran.

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

В результаті даної роботи було доведено, що центр контролю відеобладнання витримує всі задані перевантаження без пошкоджень та вимоги безпеки виконуються.

1 INTRODUCTION

Composite materials have played an important role throughout human history, from housing early civilizations to enabling future innovations. Composites offer many benefits; the key among them are corrosion resistance, design flexibility, durability, light weight, and strength. Composite materials are formed by combining two or more materials that have quite different properties, and they do not dissolve or blend into each other. The different materials in the composite work together to give the composite unique properties.

The considered composite structure of system equipment are identified as the "Cabin Electronics Compartment" (CEC) in industry standard. This structure relates to In-Flight Entertainment System. CEC controls and monitors the operation of all video systems in the passenger compartment of the aircraft.

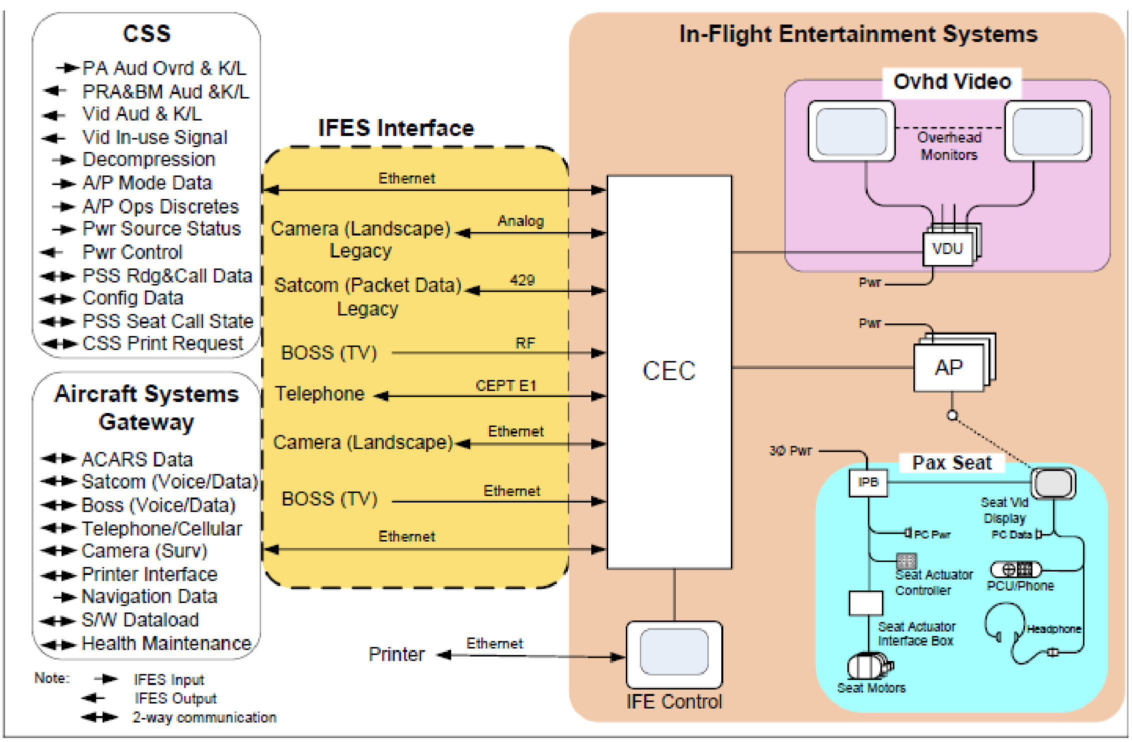


Figure 1.1. Standard interface, conceptual architecture

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. Cabin System equipment may also be installed in traditional cabin structures not originally designed for it, including galleys and lavatories.

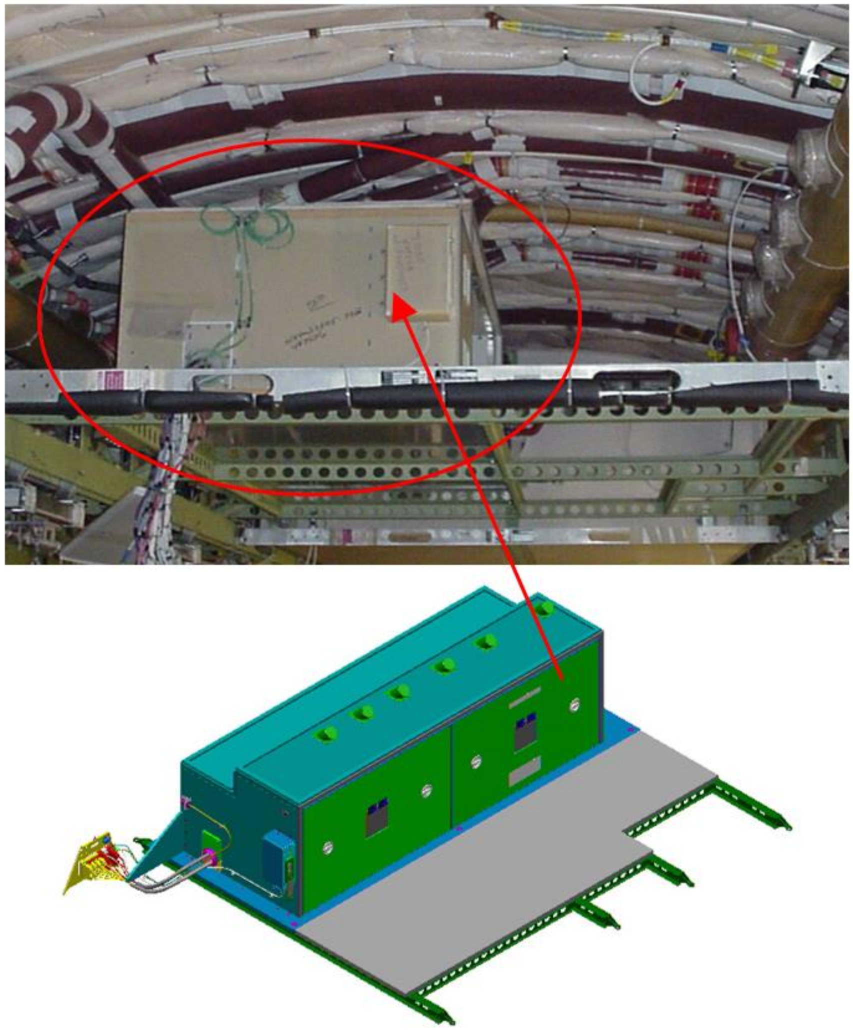


Figure 1.2. Structure definition. Photo

So, Cabin Entertainment Center (CEC) is necessary item for comfortable flight of passengers.

Figure 1.3. Flight Entertainment Systems

Therefore, it is very important that this system is protected from environmental influences in order to prevent short circuits and half-bags of expensive equipment. CEC must be safe for passengers, that is, to maintain structural integrity under operational loads and not collapse in emergency situations.

The task is to analyze the new type of CEC for compliance with safety requirements and to prove that all margins of safety are positive.

2 FINITE ELEMENT METHOD OF AEROSPACE STRUCTURE

2.1 Basic idea of finite element method

The finite element method (FEM) is a numerical technique used to perform finite element analysis (FEA) of any given physical phenomenon. It is necessary to use mathematics to comprehensively understand and quantify any physical phenomena, such as structural or fluid behavior, thermal transport, wave propagation, and the growth of biological cells. Most of these processes are described using partial differential equations (PDEs). However, for a computer to solve these PDEs, numerical techniques have been developed over the last few decades and one of the most prominent today is the finite element method.

The basic idea in the finite element method is to find the solution of a complicated problem by replacing it with a simpler one. Since the actual problem is replaced by a simpler one in finding the solution, we will be able to find only an approximate solution rather than the exact solution. The existing mathematical tools will not be sufficient to find the exact solution (and sometimes, even an approximate solution) of most of the practical problems. Thus, in the absence of any other convenient method to find even the approximate solution of a given problem, we have to prefer the finite element method. Moreover, in the finite element method, it will often be possible to improve or refine the approximate solution by spending more computational effort. The main FEM purpose 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 (or six DOF - degrees of freedom at a node): three translations and three rotations.

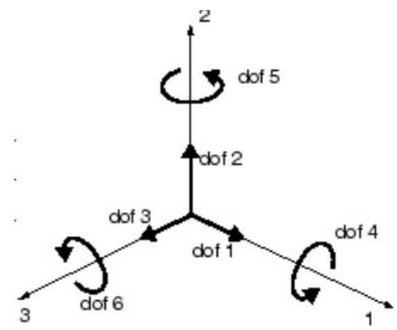


Figure 2.2. Degrees of freedom (DOF) at node.

2.2 Basic elements shapes

FEM divides some model into small pieces. Those are called Finite Elements (FE). Those Elements connect all characteristic points (called Nodes) that lie on their circumference. This “connection” is a set of equations called shape functions.

The shapes, sizes, number, and configurations of the elements have to be chosen carefully such that the original body or domain is simulated as closely as possible without increasing the computational effort needed for the solution. Mostly the choice of the type of element is dictated by the geometry of the body and the number of independent coordinates necessary to describe the system. If the geometry, material properties, and field variable of the problem can be described in terms of a single spatial coordinate, we can use the one-dimensional or line elements shown in Fig. 2.3A. The temperature distribution in a rod (or fin), the pressure distribution in a pipe flow, and the deformation of a barunderaxial load, for example, can be determined using these elements. Although these elements have a cross-sectional area, they are generally shown schematically as a line element (Fig. 2.3B). In some cases, the cross-sectional area of the element may be nonuniform.

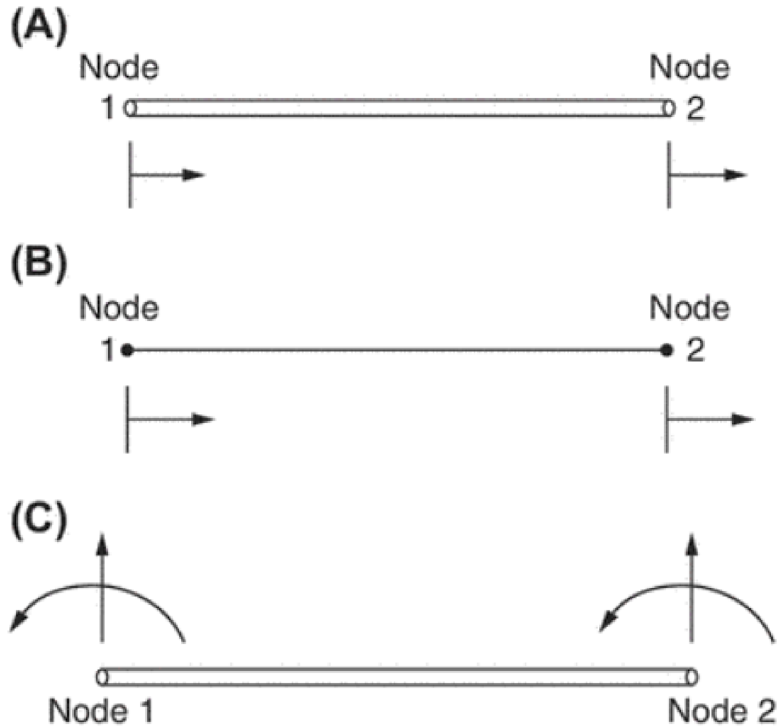


Figure 2.3 One-dimensional or line elements

For a simple analysis, one-dimensional elements are assumed to have two nodes, one at each end, with the corresponding value of the field variable chosen as the unknown (dof). However, for the analysis of beams, the values of the eld variable (transverse displacement) and its derivative (slope) are chosen as the unknowns (dof) at each node as shown in Fig. 2.3C.

When the configuration and other details of the problem can be described in terms of two independent spatial coordinates, we can use the two-dimensional elements shown in Fig. 2.4. The basic element useful for two-dimensional analysis is the triangular element. Although a quadrilateral element (or its special forms, the rectangle and parallelogram) can be obtained by assembling two or four triangular elements, as shown in Fig. 2.5; in some cases the use of quadrilateral (or rectangle or parallelogram) elements proves to be advantageous. For the bending analysis of plates, multiple dof (transverse displacement and its derivatives) are used at each node.

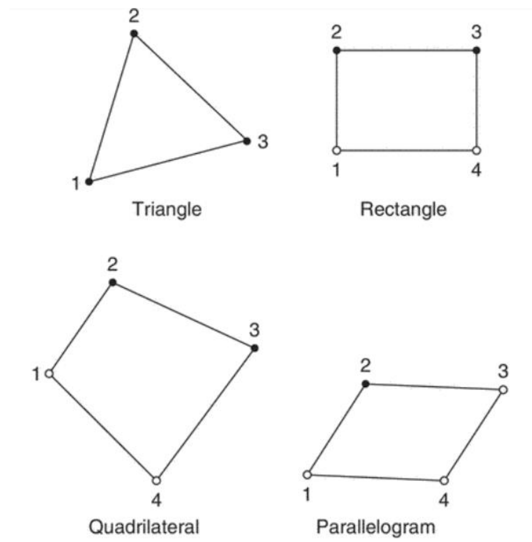


Figure 2.4. Two-dimensional elements

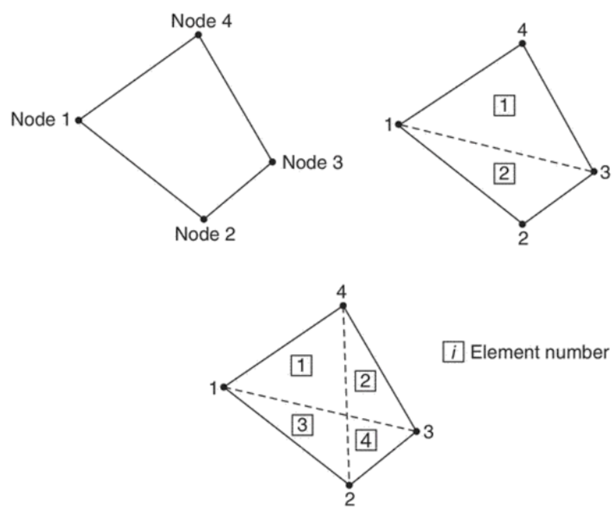


Figure 2.5. A quadrilateral element as an assemblage of two or four triangular element

If the geometry, material properties, and other parameters of the body can be described by three independent spatial coordinates, we can idealize the body by using the three-dimensional elements shown in Fig. 2.6. The basic three-dimensional element, analogous to the triangular element in the case of two-dimensional problems, is the tetrahedron element.

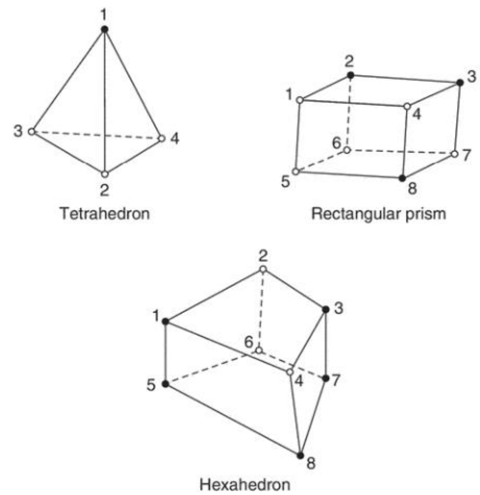


Figure 2.6. Three-dimensional elements

In some cases the hexahedron element, which can be obtained by assembling five tetrahedrons as indicated in Fig. 2.7, can be used advantageously.

Some problems, which are actually three-dimensional, can be described by only one or two independent coordinates. Such problems can be idealized by using an axisymmetric or ring type of elements shown in Fig. 2.8. The problems that possess axial symmetry, such as pistons, storage tanks, valves, rocket nozzles, and reentry vehicle heat shields, fall into this category.

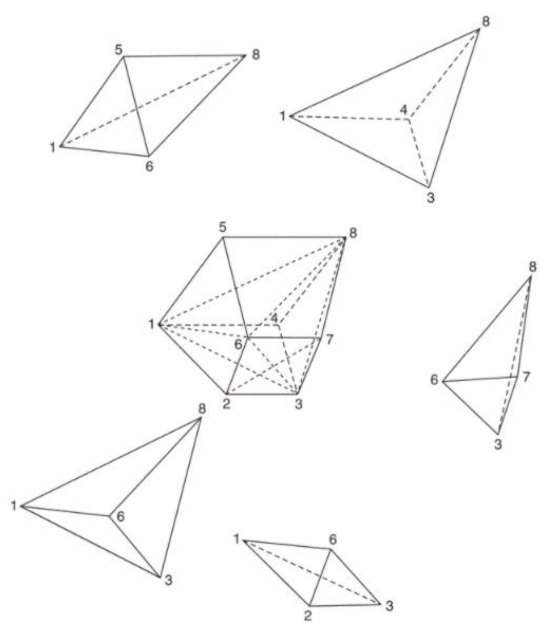


Figure 2.7 A hexahedron element as an assemblage of five tetrahedron elements.

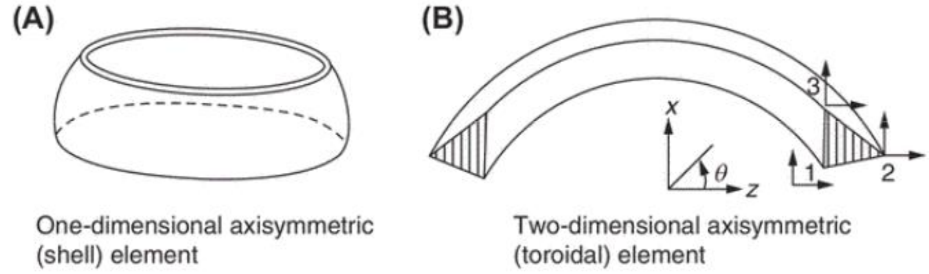


Figure 2.8. Axisymmetric elements

2.3 FEA solution

Finite element analysis (FEA) 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:

1. Pre-process or modeling the structure
2. Analysis
3. 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’s 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 selection of proper elements requires prior experience with FEA, knowledge of structure's behavior, available elements in the software and their characteristics, etc. 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,

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

Where

$\{F\}$ = External force matrix.

$[K]$ = Global stiffness matrix

$\{u\}$ = Displacement matrix

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.

2.4 Finite element procedure for composite structure

Finite element analysis consists of the following major steps:

1. A mesh encompassing the structure is generated (Fig.2.9).

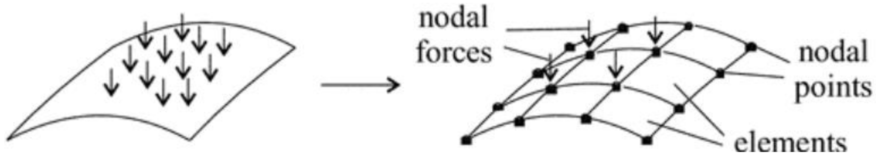


Figure 2.9. Structure and its finite element mesh

2. The stiffness matrix [k] of each element is determined.

3. The stiffness matrix [K] of the structure is determined by assembling the element stiffness matrices.

4. The loads applied to the structure are replaced by an equivalent force system such that the forces act at the nodal points.

5. The displacements of the nodal points d are calculated by

$$[K]d = f,$$

where f is the force vector representing the equivalent applied nodal forces.

6. The vector d is subdivided into subvectors δ, each δ representing the displacements of the nodal points of a particular element.

7. The displacements at a point inside the element are calculated by

$$u = [N]\delta,$$

where the vector u represents the displacements and [N] is the matrix of the shape vectors.

8. The strains at a point inside the elements are calculated by

$$\varepsilon = [B]\delta,$$

where [B] is the strain–displacement matrix.

9. The stresses at a point inside the element are calculated by

$$\sigma = [\mathrm{E}]\varepsilon,$$

where [E] is the stiffness matrix characterizing the material.

10. The element stiffness matrix, referred to in Step2, is defined as

$$[\mathrm{k}]\delta = \mathrm{fe},$$

where fe represents the forces acting at the nodal points of the element. The element stiffness matrixis is

$$[\mathrm{k}] = \int_{(v)} [B]^T [\mathrm{E}]\,[\mathrm{B}]\ \mathrm{dV},$$

where V is the volume of the element.

The preceding steps apply to structures made of either isotropic or composite materials. The only difference between isotropic and composite structures is in the material stiffness matrix [E].

3 HONEYCOMB SANDWICH STRUCTURE DEFINITION

3.1 Sandwich Terminology

Honeycomb structures are remarkable high- strength-to-weight assemblies finding broad use today in a wide range of industries as diverse as aerospace, automotive, shipbuilding, commercial equipment, and general packaging. Many honeycomb structures are made from metal, and brazing is used to create a wide variety of light-weight structures that are very strong, leak tight, and able to handle high-temperature service very well.

The sandwiched panel assembly shown in Fig. 3.1. It is comprised of a central honeycomb core and top and bottom closure panels or face sheets. The central core consists of a plurality of cells that have four- or six-sided polygonal cross sections that are brazed to the face sheets using one of the many brazing filler metals.

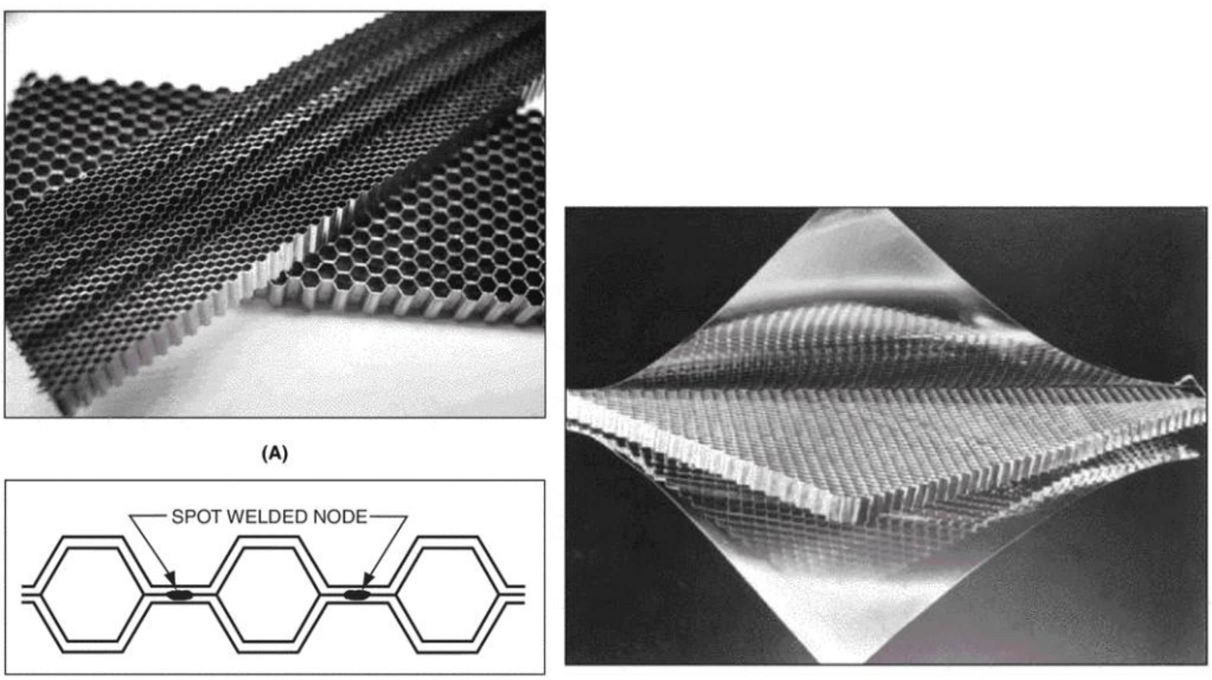


Figure 3.1. Sandwiched panel assembly. Photo

The face sheets are the prime load-bearing members. The complete stabilization of the facing surfaces by means of the proper honeycomb core design permits the desired panel strength to be attained, even when the face sheets and the honeycomb core use thin-gauge materials. The core performs the vital function of providing essentially continuous support to the face sheets by preventing buckling while at the same time transmitting shear forces. Furthermore, excellent stiffness, vibration dampening, thermal, acoustic, and insulation properties are inherent. The favorable properties of the brazed sandwich, when manufactured properly, can be maintained even at elevated temperatures.

So, a structural sandwich consists of:

- Face panels
- Core material
- Adhesives to join them together

The concept behind sandwich construction is that:

– Faces carry tensile, compressive, and bending loads.

– Cores carry shear loads.

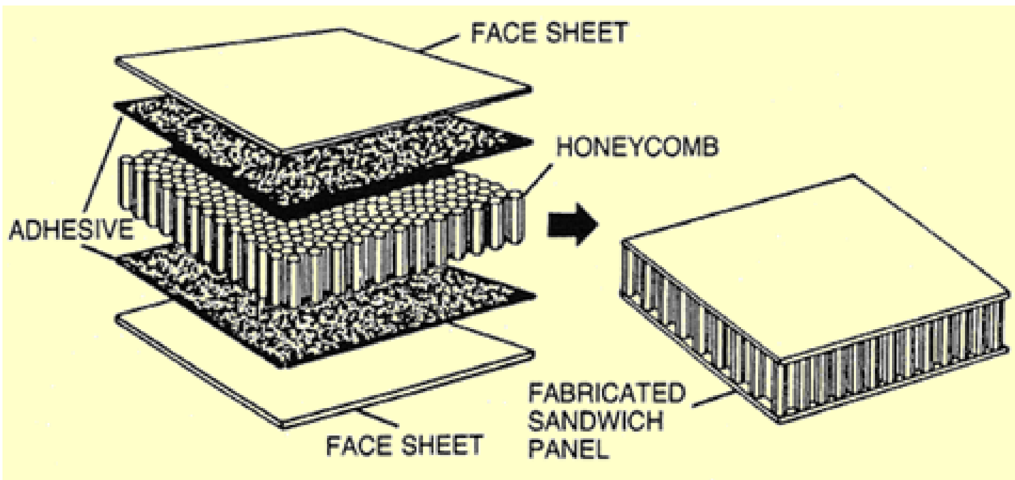


Figure 3.2. Sandwich construction

Face materials

- Typically, composite panels.
- Other face materials can include aluminum or other metals or plastic sheets.
- Typically, 0.01 to 0.5 inches (0.3 to 13mm) thick.

Chosen on the basis of:

- Weight
- Strength
- Fabricability

Adhesives

Chosen for:

- Giving strength

- Temperature compatibility
- Ability to form a fillet at the cell wall or in some other way to form a good bond with the minimal surface area of the end of some of the core materials.

Core materials

Can be made of many materials, most commonly:

- Wood
- Rigid foam
- Honeycomb (aluminum, Nomex, phenolic, Kevlar, carbon, fiberglass)

Typical densities of core materials are 1 – 55 lb/ft3 (.016 to .88 g/cm3).

3.2 Analysis details for honeycomb sandwich panels

Basic properties of composites:

•E11, E22.

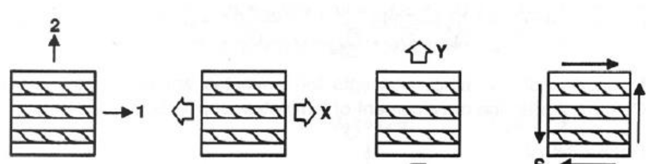
•G12, ν12.

Material Allowables:

•X (Tension/compression in direction 1).

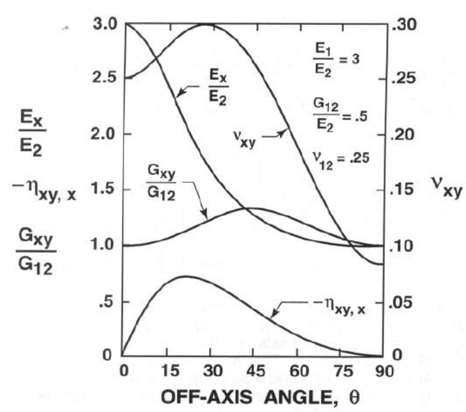
•Y (Tension/compression in direction 2).

•S (In-plane Shear).

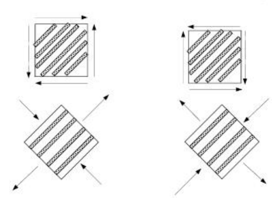


Fundamental Strengths for a Unidirectionally Reinforced Lamina

X_t = axial or longitudinal strength in tension
X_c = axial or longitudinal strength in compression
Y_t = transverse strength in tension
Y_c = transverse strength in compression
S = shear strength



Normalized Moduli for Glass-Epoxy



Effect of Shear Load

Figure 3.3. Composite Strength

3.3 Failure modes of Composite structure

Panel safety margin calculation depends on the mode of failure. Mode of failure can be divided in to two groups:

1. Strength Based failure modes;
2. Stability Based Failure modes.

1. Strength Based Failure modes:
   - Face sheet failure
   - Transverse shear failure (core shear)
   - Flexural core crushing
   - Flat wise Tension or compression
2. Stability Based Failure Modes:
   - Panel Buckling.
   - Face Wrinkling
   - Face sheet Dimpling or Intra cell buckling
   - Shear Crimping

Facesheet Strength Failure Details

1. Facing failure is simply characterized by cracked facesheets. This failure occurs when the facesheet strength is exceeded.

2. Transverse shear failure can manifest itself as face-to-core debonding or as a shear failure in the core itself. This failure occurs when the core or the face-to-core adhesive has insufficient shear strength.

3. Flexural core crushing is a concern when the facesheets tend to move towards each other under the influence of bending. This failure mode occurs when the core has insufficient compression strength.

4. Flatwise tension or compression occurs in the ramp area where the bag-side facesheet changes direction. The flatwise, or interlaminar, stresses are induced at the ramp radii. A flatwise tension stress can cause face-to-core debonding, while a flatwise compression stress can cause core crushing.

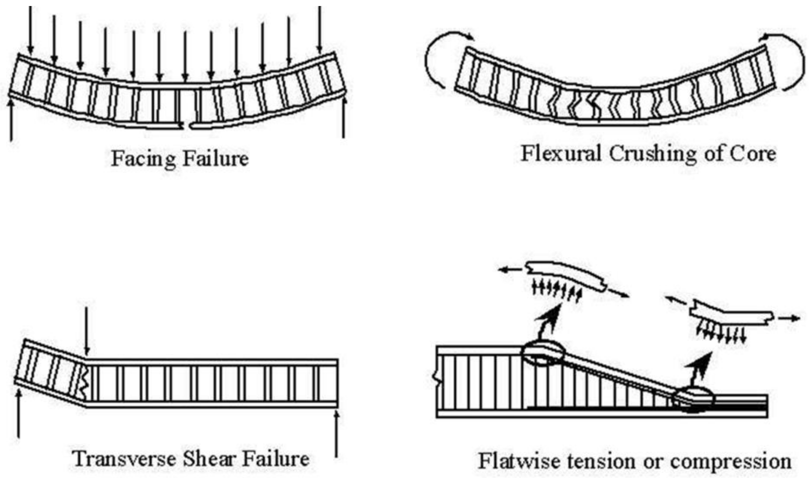


Figure 3.3. Facesheet Strength Failure

Local Instability Failure Modes:

1. Intracell buckling or face dimpling is a local instability characterized by the buckling of a facesheet into or out of the confines of a single cell. This failure can occur when the facesheets are thin.

2. Face wrinkling is a local instability characterized by the inward or outward buckling of the face, accompanied by core crushing, core tearing, or face-to-core debonding. This failure can occur when the core has a low density.

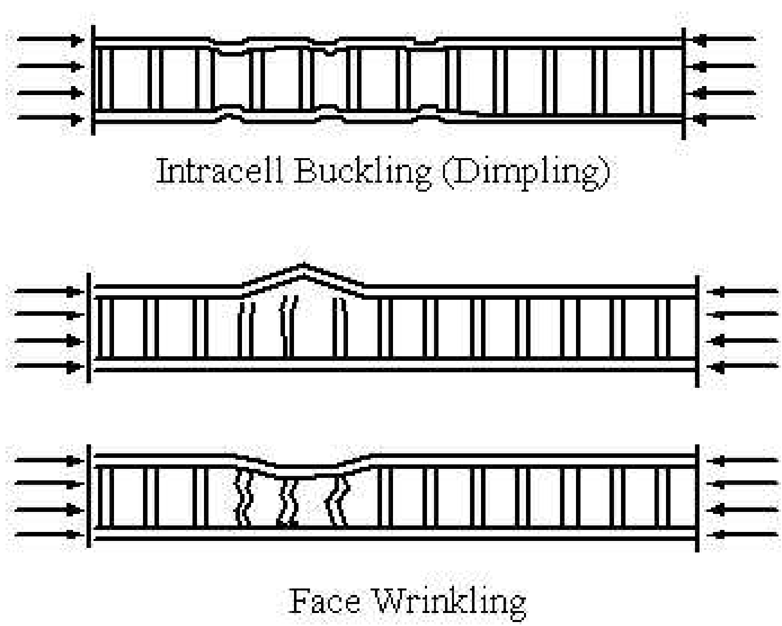


Figure 3.4. Local Instability

3.4 Failure Criteria for face sheet failure Mode

Several failure criteria have been developed based on the strength of each lamina in a laminate. When applied to laminates these criteria predict the failure based on the first ply failure and a progressive ply-by-ply failure analysis needs to be performed to predict the laminate failure more accurately.

Lamina based Failure theories:

- Maximum Stress Criteria
- Maximum Strain Criteria
- Tsai-Hill Criterion
- Hoffman's Criterion
- Tsai-Wu Criterion
- Strain Invariant Failure Theory (SIFT)
- And other numerous theories.

However, Tsai-Hill, Tsai-Wu, Hoffman and other similar failure criteria are not recommended for analyzing the face sheet strength, because all these criteria are based on lamina strength where as our allowables are based on laminate strength. (Lamina strength does not account for lamina interaction within laminate in failure criteria).

Proposed Method for Calculating Strength MS:

Use the following criteria for calculating the Margin of Safety3

Case 1. When both S11 and S22 are tension and S11 >S22 no shear

$$M_s = \frac{X_T}{S_{11}} - 1$$

Case 2. One tension and One compression (i.e. S11 > and S22 <0) no Shear

$$M_s = \frac{1}{\dfrac{S_{11}}{X_T} + \dfrac{|S_{22}|}{|X_C|}} - 1$$

Case 3. Tension or compression with the shear

$$M_s = \frac{1}{\sqrt{(\frac{S_{11}}{X_T})^2 + (\frac{S_{12}}{X_s})^2}} - 1$$

Where: S11, S22, S12 are stresses in fiber direction and XT, Xc and Xs are design values in Tension, Compression and Shear respectively.

Above failure criteria are applicable for woven fabric assuming strength and stiffness properties in “1” and “2” directions are equal (or nearly equal).

3.5 Analysis Assumptions and Benefits

Assumptions:

1. The failure criteria for face sheet and core are not identical.

2. PCL has been developed only for honeycomb sandwich panels with composite properties defined as exactly a 3-layer laminate whereby:

-Layer 1 is an equivalent face sheet representing one or more plies

-Layer 2 is the core

-Layer 3 is an equivalent face sheet representing one or more plies

If composite elements are not defined with this exact 3-layer laminate format this PCL will generate incorrect results.

Margins of Safety calculated at 5 or higher will be displayed as exactly 5 in the results.

Benefits:

1. Implementation of SMA failure criteria’s for “Honeycomb Sandwich Panels” in MSc Patran

2. Reduction in analysis time and manual errors

3. Visual plot for Margin of Safety (All fringe plot options can be used during post-processing)

4. Detailed analysis and plots for sandwich panels (like layer by layer results, failure index values and plots, critical layer plot etc).

3.6 The method of calculation margin of safety for Honeycomb Sandwich Panels

The method of calculation margin of safety for Honeycomb Sandwich Panels using SMA failure criteria has been automated with the help of Patran Command Language (PCL). This PCL uses "User Defined" failure criteria option available in "MSc. Patran - Laminate modeler" module and failure criteria used for calculation of MS is as follows:

For Face sheet:

When S11 and S22 are both positive (tension) or negative (compression):

For both tension:

- Critical component - layer ID-1

$$MS = \frac{1}{\sqrt{\left(\frac{S11}{X1t}\right)^2 + \left(\frac{S22}{X2t}\right)^2 + \left(\frac{S12}{Xs}\right)^2}} - 1$$, where S – action stress; X –allowable stress

For both compression:

- Critical component - layer ID-2

$$MS = \frac{1}{\sqrt{\left(\frac{S11}{X1c}\right)^2 + \left(\frac{S22}{X2c}\right)^2 + \left(\frac{S12}{Xs}\right)^2}} - 1$$, where S – action stress; X –allowable stress

S11 and S22 has different sign and S12=0:

For S11 tension and S22 compression:

- Critical component - layer ID-3

$$MS = \frac{1}{\left(\frac{S11}{X1t}\right) + \left(\left|\frac{S22}{X2c}\right|\right)} - 1$$, where S – action stress; X –allowable stress

For S11 compression and S22 tension:

- Critical component ID-3

$$MS = \frac{1}{\left(\left|\frac{S11}{X1c}\right|\right) + \left(\frac{S22}{X2t}\right)} - 1$$, where S – action stress; X –allowable stress

S11 and S22 has different sign but S12 $\neq$ 0:

a) S11 tension and S22 compression:

- Critical component ID-4

$$MS = \min\left(\frac{1}{\sqrt{\left(\frac{S11}{X1t}\right)^2 + \left(\frac{S12}{Xs}\right)^2}} - 1;\ \frac{1}{\sqrt{\left(\frac{S22}{X2c}\right)^2 + \left(\frac{S12}{Xs}\right)^2}} - 1;\ \frac{1}{\left(\frac{S11}{X1t}\right) + \left(\left|\frac{S22}{X2c}\right|\right)} - 1\right),$$

where S – action stress; X –allowable stress

b) S11 compression and S22 tension:

- Critical component ID-4

$$MS = \min \left(\frac{1}{\sqrt{\left(\frac{S11}{X1c}\right)^2 + \left(\frac{S12}{Xs}\right)^2}} - 1, \frac{1}{\sqrt{\left(\frac{S22}{X2t}\right)^2 + \left(\frac{S12}{Xs}\right)^2}} - 1; \frac{1}{\left(\frac{S11}{X1c}\right) + \left(\left|\frac{S22}{X2t}\right|\right)} - 1\right),$$

where S – action stress; X –allowable stress

For CORE:

- Critical component ID-5

$MS = \min \left(\frac{X13}{S13} - 1; \frac{X23}{S23} - 1\right)$, where S – action stress; X –allowable stress

4 CABIN ENTERTAINMENT CENTER LOCATION AND STRUCTURE

Cabin Entertainment Center is installed in the crown area above the cabin ceiling and attached to the ceiling beams of the aircraft via seven Shear ties.

Total weight of the following Cabin Entertainment Center (CEC) components is 359.73 lbs.

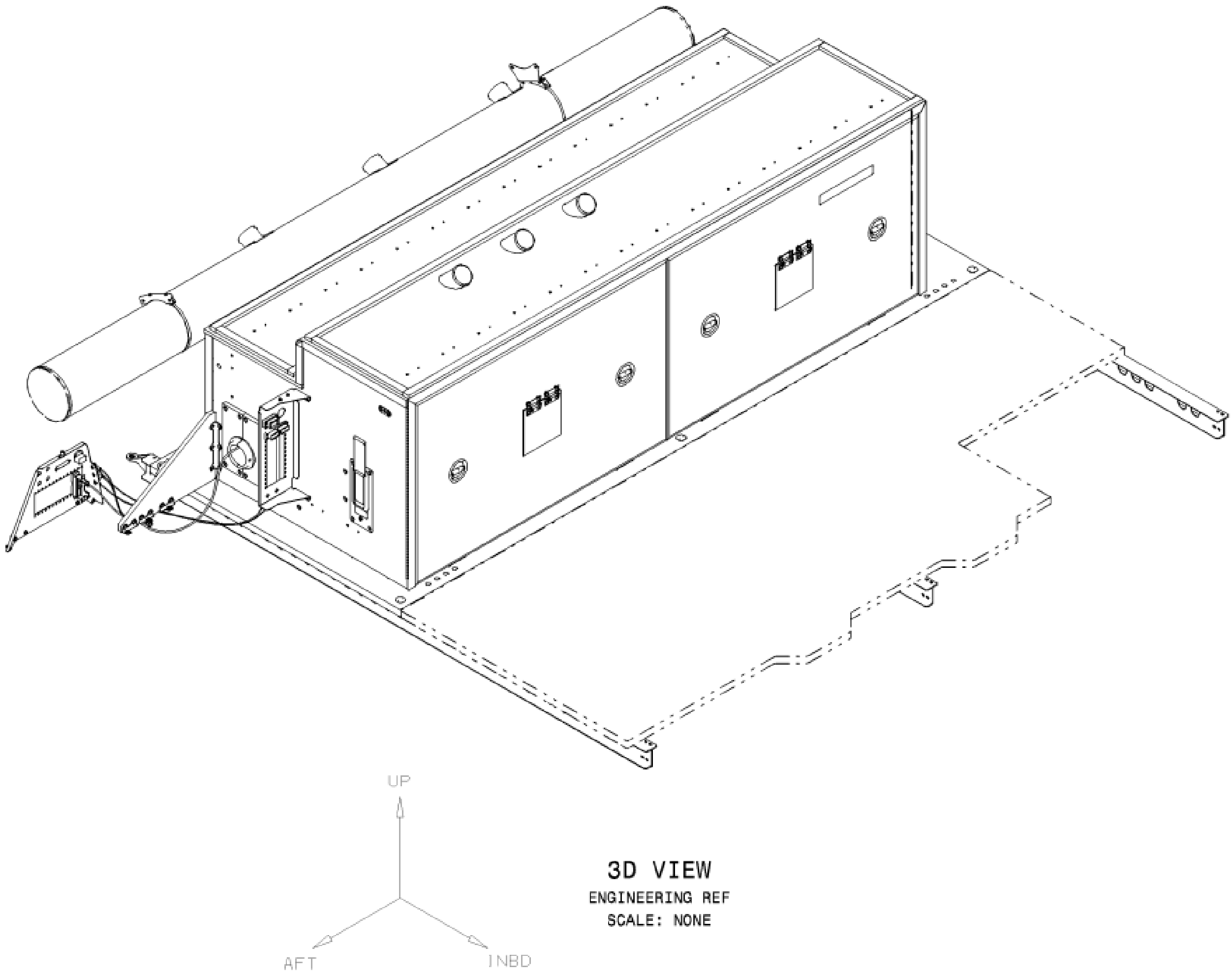


Figure 4.1. CEC Installation

The panels are honeycomb sandwich panels with face sheets with a thickness of either 1 inch, 0.50 inch and 0.55 inch. The Cabin Entertainment Center structure is shown in Figure 4.2.

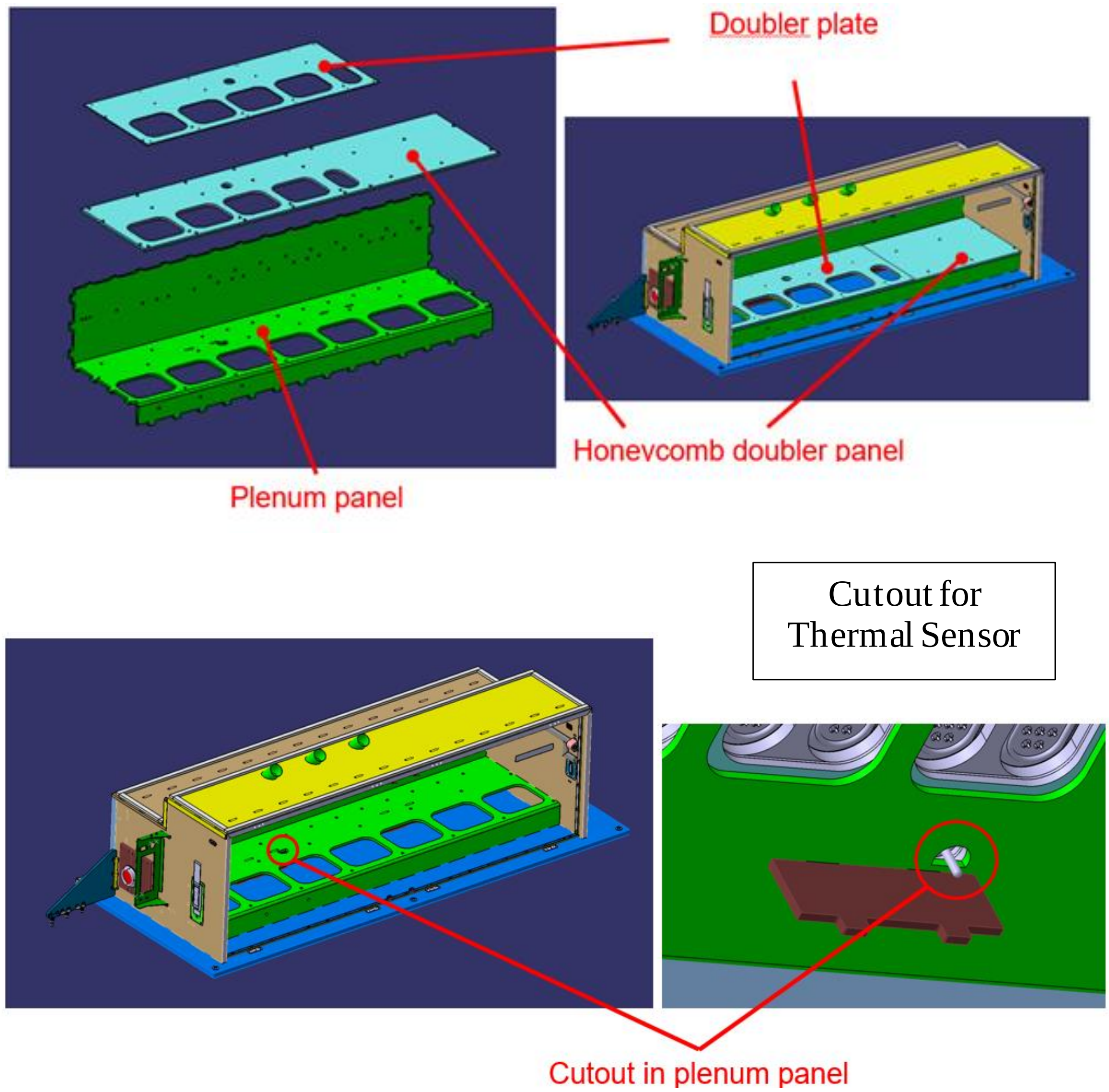


Figure 4.2. CEC structure

5 JOINT DEFINITION

The CEC Panels are jointed to each other using Bolts, Inserts, Dog-Bones and Tab and Slot connections.

5.1 Tab and Slot Joint

An important consideration in using any tab and slot joint is that the geometry must represent the as at as-tested conditions to the greatest extent possible. This is necessary in order to provide the strength as prescribed by the design allowable. One key parameter is the pitch (or spacing) in between tabs. The pitch needs to represent the minimum width of coupon used in testing.

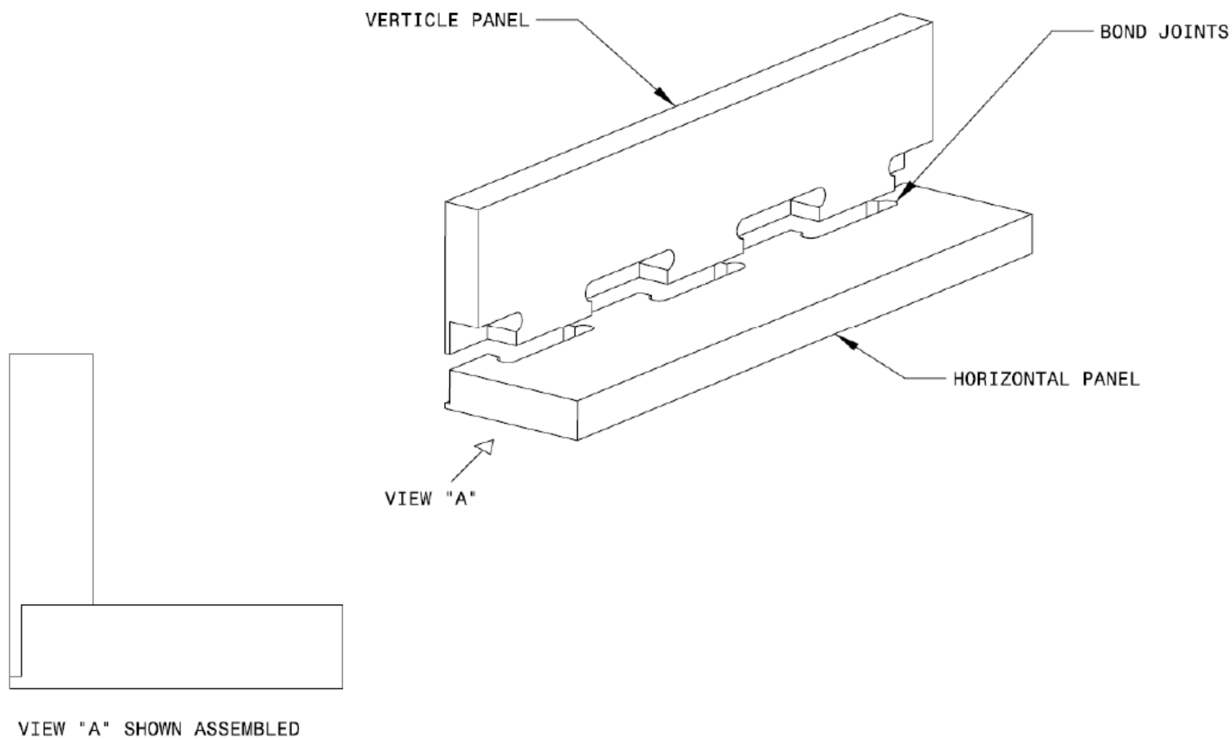


Figure 5.1. Tab-slot construction

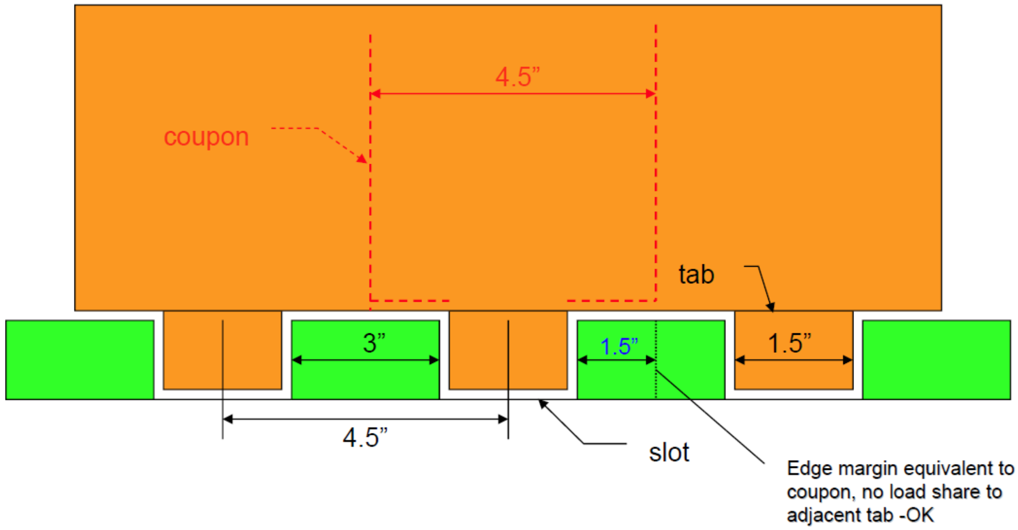


Figure 5.2. Tab-slot construction

Failure mode:

a) Plug disbonding from bottom facesheet and core.

b) Core shear.

c) Facesheet disband from core.

5.2 Dog-Bones joints

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 5.3.

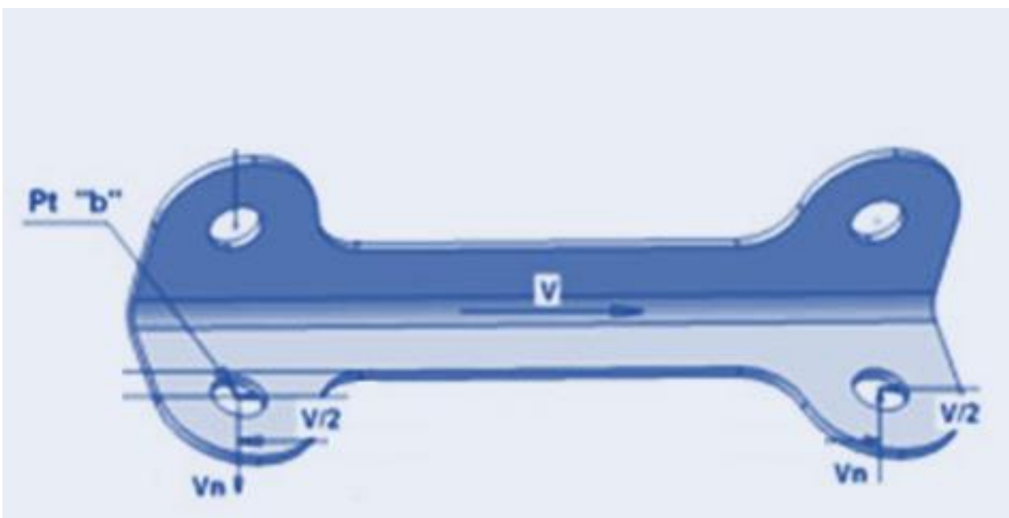


Figure 5.3. Dog-Bones joints

5.3 Bolts and Inserts

Inserts have been studied since sandwich panels became popular for weight reduction applications. As there are many different uses for panels, the inserts have been designed accordingly. As result, they are made in several geometries, materials, for different load conditions, etc. They are commonly made of aluminum 2024 alloy. The bolted joint with Inserts is presented in the Figure 5.4.

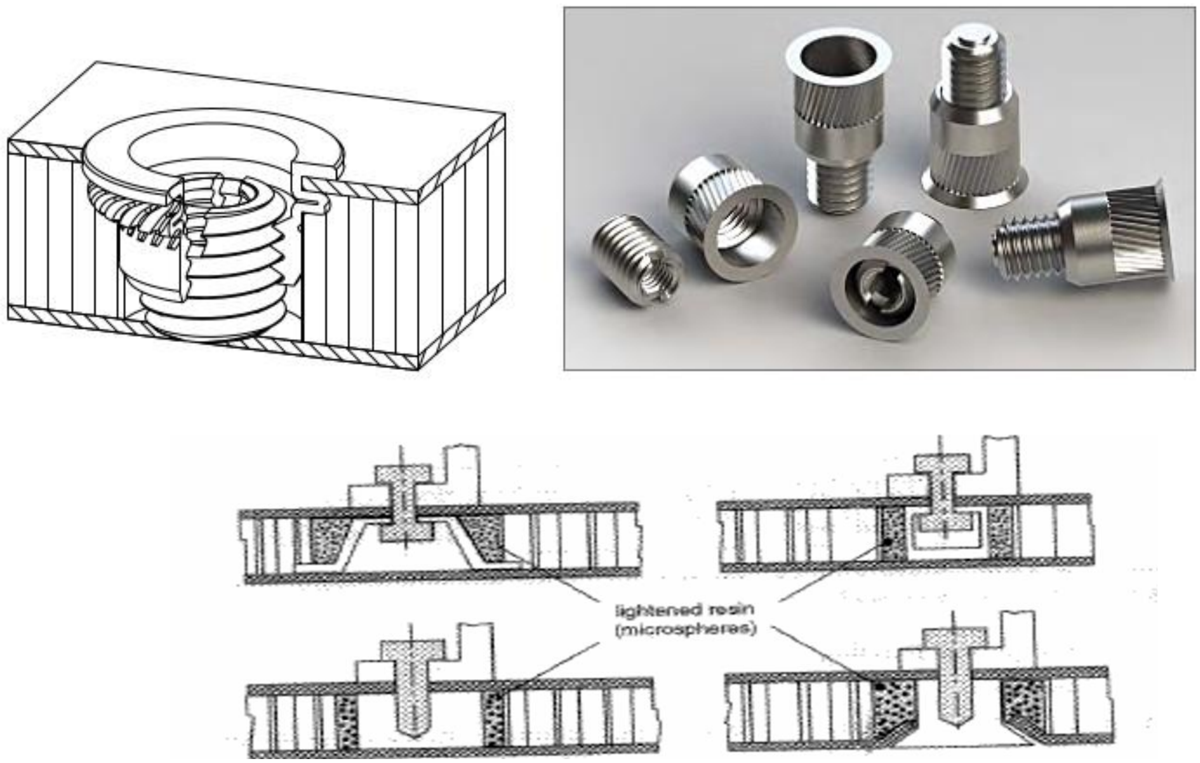


Figure 5.4. Inserts in Sandwich Construction

5.3 Bonded joint

The bonded joint is presented in the Figure 5.5.

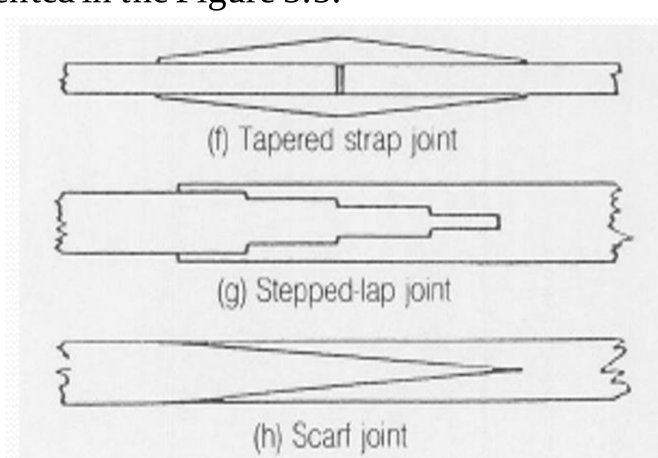


Figure 5.5. Bonded joints

6 PROPERTIES AND DESIGN VALUES

- All the material properties such as E11, E22 and G12 are provided in this material coordinate system, and they are sensitive to referencing system.
- Most composite analyses are based on stresses in the principle material direction. If the material orientation is not known, i.e., θ is unknown, no credible analysis can be done.

The material properties and design values used in the analysis are presented in this section. The structural panels used in the construction of the CEC1 are described in Table 6.1.

Table 6.1 – CEC Panel Assy Description

Panels	Thickness (in)	Core*	Facesheet*
Panel Assy – Stand Off For Wire Bundle	0.55	24	A (2/2)
Panel Assy – Floor	1.00	26	A (2/2)
Panel Assy – Fwd	1.00	26	A (2/2)
Panel Assy – Aft	1.00	26	A (2/2)
Panel Assy – Back	1.00	26	A (2/2)
Panel Assy – Plenum Support	0.5	48	A (2/2)
Panel Assy – Plenum	0.5	48	A (2/2)
Panel Assy – Filter	0.5	48	A (2/2)
Panel Assy – Ceiling	1.00	26	A (2/2)
Panel Assy – CEC Doubler Panel	0.5	27	B (1/1)

Notes: * Parenthesis denotes number of plies (2/2: 2 plies per side);

Facesheet A: 0.011in thick;

Facesheet B: 7075-T6 Bare sheet, 0.032 in thick;

Core 26: 0.95 in thick;

Core 48: 0.47 in thick;

Core 27: 0.436 in thick;

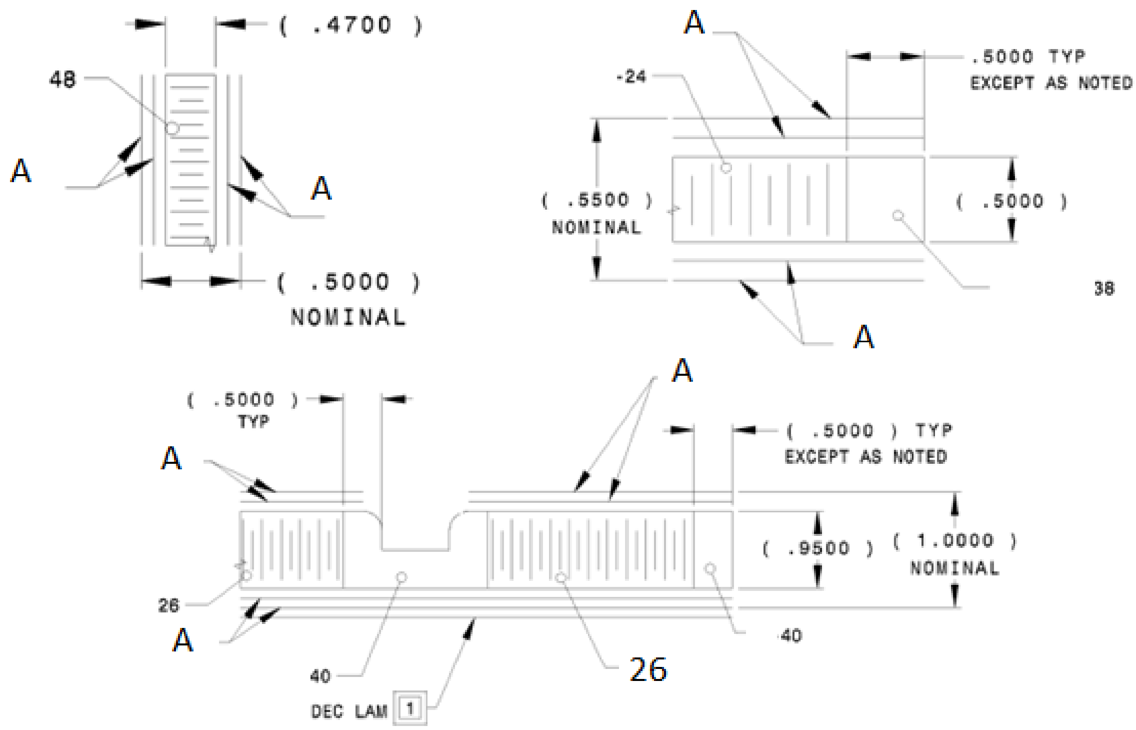


Figure 6.1. Panel Build-Up, Fiberglass Facesheets

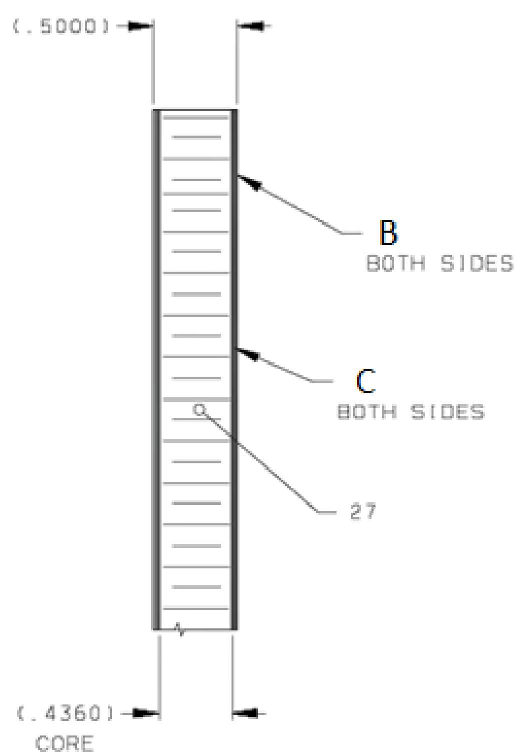


Figure 6.2. Panel Build-Up, Aluminum Facesheets

Table 6.2 – Sandwich Panel Material Properties

Ply	Identification	Tens./Comp. Modulus (psi)		Shear Modulus (psi)			Poisson Ration
		E11	E22	G12	G23	G13	
Face sheet	A (MODIFIED PHENOLIC PREIMPREGNATED GLASS FABRIC FOR INTERIOR SANDWICH PANEL AND LAMINATES)	3146500	3146500	680000	-	-	0.13500001
Core	24 (48) (NONMETALLIC HONEYCOMB CORE), 0.5 in thk.	-	-	-	3700	6900	0.13
	26 (NONMETALLIC HONEYCOMB CORE), 0.95 in thk.	-	-	-	3696.3	6382.5	

Table 6.3 – Sandwich Panel Design Values

Type	**Face Sheet (# of plies) / Core**	**Facesheet Tension / Compression Failure (ksi)**	**Facesheet In-Plane Shear (psi)**	**Core Shear Failure (psi)**
0.5 in	A (2) (MODIFIED PHENOLIC PREIMPREGNATED GLASS FABRIC FOR INTERIOR SANDWICH PANEL AND LAMINATES) / 24(48) (NONMETALLIC HONEYCOMB CORE)	35/26	7432	86
1.0 in	A (2) (MODIFIED PHENOLIC PREIMPREGNATED GLASS FABRIC FOR INTERIOR SANDWICH PANEL AND LAMINATES) / 26 (NONMETALLIC HONEYCOMB CORE)	35/26	7432	76
0.5 in	B (1) (7075 T6 Bare Sheet 0.032) / 27 (NONMETALLIC HONEYCOMB CORE)	81/81	27500	86

Tab and Slot Design Values

Table 6.4 – Tab and Slot Joint Allowables

Joint Type	Aux	T/S Type	Tab Panel	Slot Panel	Edge Margin(in)	Tab Length(in)	Pt (lb)	Pis lb)	Ps (lb)	Mb (lb*in)
A	1-1-Long	IMT	Panel 1.0 in	Panel 1.0 in	>3"	1.5	814	1084	329	636
B	1-1-Short	IMT BCh	Panel 1.0 in	Panel 1.0 in	0.25"	1.5	380	978	355	660
C	1-0.5-Short	IMT BCh	Panel 1.0 in	Panel 0.5 in	0.25"	1.5	149.4	734	232	99.6
D	0.5-1-Long	IMT	Panel 0.5 in	Panel 1.0 in	>3"	1.5	488	804	322	222
E	0.5-0.5-Long	IMT	Panel 0.5 in	Panel 0.5 in	>3"	1.5	488	804	322	222
F	1-0.5-Long	IMT	Panel 1.0 in	Panel 0.5 in	>3"	1.5	488	804	322	222

Insert design values are tabulated below.

Table 6.5 – Insert Shear Design Values

Part	Core Thickness (in)	Core Material	Facesheet	Ply / Face	ED	B-Design Value (lbs)
Insert A3	0.5	NONMETALLIC HONEYCOMB CORE	MODIFIED PHENOLIC PREIMPREGNATED GLASS FABRIC FOR INTERIOR SANDWICH PANEL AND LAMINATES	2	1.5	525
	0.5	NONMETALLIC HONEYCOMB CORE	0.32 thick 7075 T6 alumin	1	0.5	1193
					1.5	1930
Insert NAS	0.5	NONMETALLIC HONEYCOMB CORE	0.32 thick 7075 T6 alumin	1	0.5	1241
					1.5	2101
Insert 4C/16C	1.0	NONMETALLIC HONEYCOMB CORE	MODIFIED PHENOLIC PREIMPREGNATED GLASS FABRIC FOR INTERIOR SANDWICH PANEL AND LAMINATES	2	N/A	760

Table 6.6 – Insert Tension Design Values

Part	Core Thickness (in)	Core Material	Facesheet	Ply / Face	B-Design Value (lbs)
Insert A3	0.5	NONMETALLIC HONEYCOMB CORE	MODIFIED PHENOLIC PREIMPREGNATED GLASS FABRIC FOR INTERIOR SANDWICH PANEL AND LAMINATES	2	177
	0.5	NONMETALLIC HONEYCOMB CORE	0.32 thick 7075 T6 alumin	1	516
Insert NAS	0.5	NONMETALLIC HONEYCOMB CORE	0.32 thick 7075 T6 alumin	1	605
Insert 4C/16C	1.0	NONMETALLIC HONEYCOMB CORE	MODIFIED PHENOLIC PREIMPREGNATED GLASS FABRIC FOR INTERIOR SANDWICH PANEL AND LAMINATES	2	760

Metal Properties:

Metal material properties for Aluminum 7075-T6 Bare Sheet are presented below:

Allowable material tensile ultimate stress: Ftu = 80 [ksi]

Allowable material tensile yield stress: Fty = 71 [ksi]

Allowable material compression yield stress: Fcy = 75 [ksi]

Allowable material shear ultimate stress: Fsu =42 [ksi]

Allowable material bearing ultimate stress: Fbru = 144 [ksi, e/d = 2]

Allowable material bearing yield stress: Fbry =112 [ksi, e/d =2]

Elastic modulus for tension: E =10.3E6 [psi]

Elastic modulus for compression: Ec = 10.5E6 [psi]

Shear modulus: G = 3.9E6 [psi]

Poisson Ration: M = 0.33.

Fastener Properties:

Table 6.8 – Fastener Design Values

Type	**Joint Description**	**Allowable**	
		P_{ten}, lbs	**P_{sh}, lbs**
BOLT, 100 DEG HEAD, CROSS RECESS, 95 KSI SHEAR	Doubler Panel to the Plenum	2600	2690

7 DESIGN LOADS

The Design Load Conditions for CEC is the Ultimate Inertial (Emergency, Flight & Ground). Loads are applied in Global CS (Figure .1).

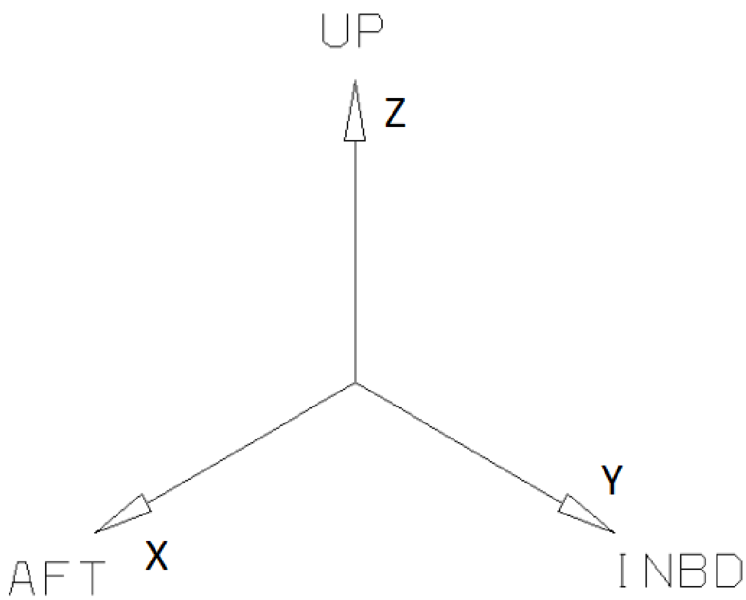


Figure 7.1. CEC Equipment Installation

FAA Ultimate Load Factors:

FWD	AFT	UP	DOWN	SIDE ①	SIDE① + 1.5G DOWN	UP + 0.8G FWD	UP + 0.5G FWD	UP + 0.5G AFT	DOWN +1.5G FWD	DOWN + 0.8G FWD	DOWN + 0.5G FWD	DOWN + 0.5G AFT	DOWN + 1.5G AFT
9.0	1.5	3.0	6.0	3.0	1.1	1.5	2.4	2.4	3.8	3.9	7.0	7.0	3.0
9.0	1.5	3.0	6.0	3.0	1.6	1.5	1.5	1.5	3.8	5.6	6.2	6.2	3.0
9.0	1.5	3.0	6.0	3.0	1.7	1.5	2.7	2.7	3.8	5.9	7.1	7.1	3.0

Figure 7.2. FAA Ultimate Load Factors

Flight and Ground Load Factors:

UP	DOWN	SIDE① + 1.5G DOWN	UP + 0.8G FWD	UP + 0.5G AFT	DOWN + 1.5G FWD	DOWN + 0.8G FWD	DOWN + 0.5G AFT	DOWN +1.5G AFT
2.1	5.1	1.1	1.5	1.5	3.8	3.9	3.9	3.0
2.6	5.6	1.6	1.5	1.5	3.8	5.6	5.6	3.0
2.8	5.8	1.7	1.5	1.5	3.8	5.9	5.9	3.0

Figure 7.3–Ultimate Flight Load Factors

Handling and Abuse loads

No Step/Abuse load is applied to CEC Installation because it is not accessible to passengers or crew members.

Decompression Loads

As the CEC Assy construction is not changed and decomp.load are the same not analysis required.

Table 7.1 – Load Factor Summary

Emergency Landing Load Factor	**Flight Load Factor**	**#**	**Applicable Load Factor Selected for Analysis***
9.0G-Fwd		1	9.0G-Fwd
1.5G-Aft		2	1.5G-Aft
3.0G-Up	2.6G-Up	3	3.0G-Up
6.0G-Dwn	5.6G-Dwn	4	6.0G-Dwn
3.0G-Right		5	3.0G-Right
3.0G-Left		6	3.0G-Left
1.6G-Right+1.5G-Dwn	1.6G-Right+1.5G-Dwn	7	1.6G-Right+1.5G-Dwn
1.6G-Left+1.5G-Dwn	1.6G-Left+1.5G-Dwn	8	1.6G-Left+1.5G-Dwn
1.5G-Up+0.8G-Fwd	1.5G-Up+0.8G-Fwd	9	1.5G-Up+0.8G-Fwd
1.5G-Up+0.5G-Fwd			(covered with previous)
1.5G-Up+0.5G-Aft	1.5G-Up+0.5G-Aft	10	1.5G-Up+0.5G-Aft
3.8G-Dwn+1.5G-Fwd	3.8G-Dwn+1.5G-Fwd	11	3.8G-Dwn+1.5G-Fwd
5.6G-Dwn+0.8G-Fwd	5.6G-Dwn+0.8G-Fwd	12	5.6G-Dwn+0.8G-Fwd
6.2G-Dwn+0.5G-Fwd		13	6.2G-Dwn+0.5G-Fwd
6.2G-Dwn+0.5G-Aft	5.6G-Dwn+0.5G-Aft	14	6.2G-Dwn+0.5G-Aft
3.0G-Dwn+1.5G-Aft	3.0G-Dwn+1.5G-Aft	15	3.0G-Dwn+1.5G-Aft

NO DOWN Load Cases

DOWN ONLY Load Cases

8 FEM ANALYSIS

Analysis Criteria for aircraft structure:

- Must be able to withstand specific impact damage requirements and still be capable of sustaining ultimate load.
- Honeycomb panels, spars and ribs must be stable to ultimate load.
- Aerodynamic smoothness: deflection at 1G cruise conditions must meet aero groups requirements.

The stress analysis of the Cabin Entertainment Center installation has been performed. MSC Patran 2017.1 code was used in the FE model development, pre-and post-processing. The MSC Nastran 2017.1 was used for the FE solver.

The finite element model of the CEC has been developed per the guidelines set forth in the Boeing Nastran/Patran Standards as posted in IRC web site.

Hand analysis has been performed as needed per standard methods and procedures.

FEM Modeling composite materials:

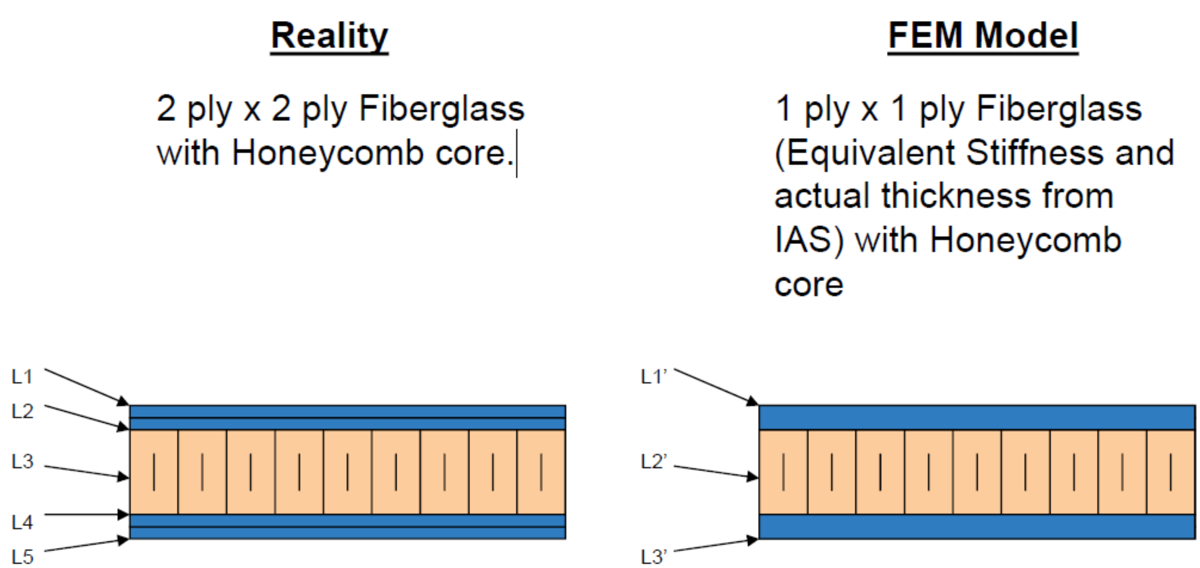


Figure 8.1 FEM Modeling of composite

FEM model name: ARO_CEC _MPC.db, saved in current stress package. The general modeling methods used are listed below:

1. Mass is represented either by lumped mass elements located at the c.g. of certain sub-structure (RBE3's are used to attach mass elements to the structure).

2. For the sandwich panels, mass is represented by non-structural mass property.

3. Insert and fastener connections are represented by a universal CBUSH element with 3 translational DOF and Y- direction vector <1 1 1>. All insert bush elements are modeled with GRID 1 at the panel, so that BUSH element report forces are positive for tension direction.

4. Tab and slots joints are modeled using SmartBush Tool 1.10.

5. It is assumed that places where cutouts are closed to the Tab and Slot (T-S) joints (Figure) these joints are weak can fail. Therefore BUSH elements of these T-S joints were removed and the FE-model became in Fail Safe mode.

6 The CEC has large contact surface between Plenum and Doubler Panel (Figure). To transmit vertical loads correctly between these parts two bdf-files (Ref. Appendix) was generated with different models for two load sets: "No Down" load case set and "Down Only".

"No Down" load case set combines load cases number 1, 2, 3, 5, 6, 7, 8, 9, 10 which has primary load in forward, aft, side or up direction . The insert and fastener connections between Doubler Panel and Plenum are represented by a universal CBUSH elements with 3 translational DOF.

"Down Only" load case set combines load cases number 4, 11, 12, 13, 14, 15 which has primary down load direction. CBUSH elements between Doubler Plate and Plenum has only 2 translational DOF in shear direction (X and Y in Global CS) in zone under equipment, and the axial Z load transferred by RBE3 elements which distribute axial load on close area around the fastener point location.

Properties of RBE3-elements for Z-load transferring presented .

7. Boundary conditions of the CEC FE-model.

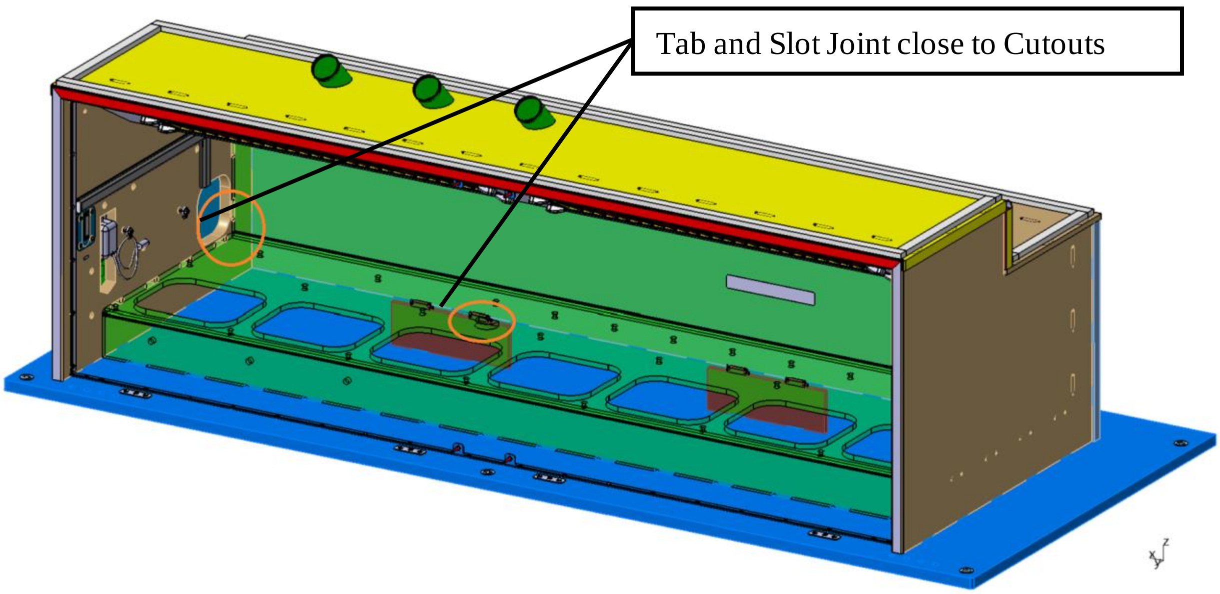


Figure 8.2. Cutouts close to T-S Joints

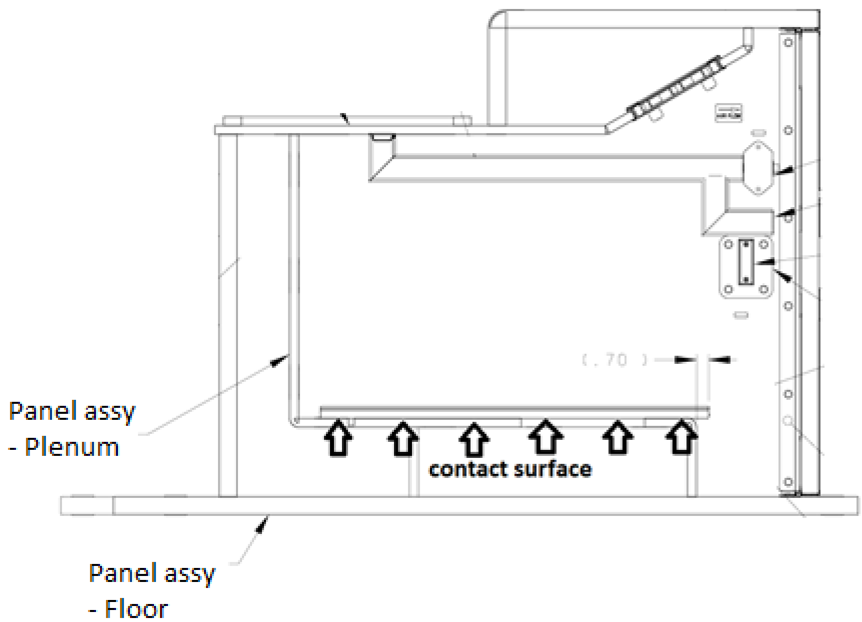


Figure 8.3. Plenum and Doubler Panel Contact Surface.

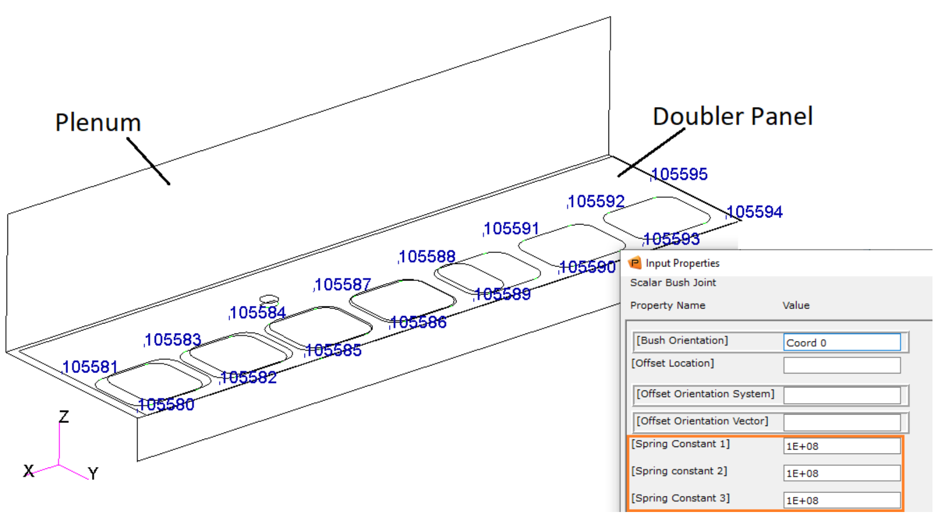


Figure 8.4. Plenum and Doubler Panel connection for “No Down” load case set

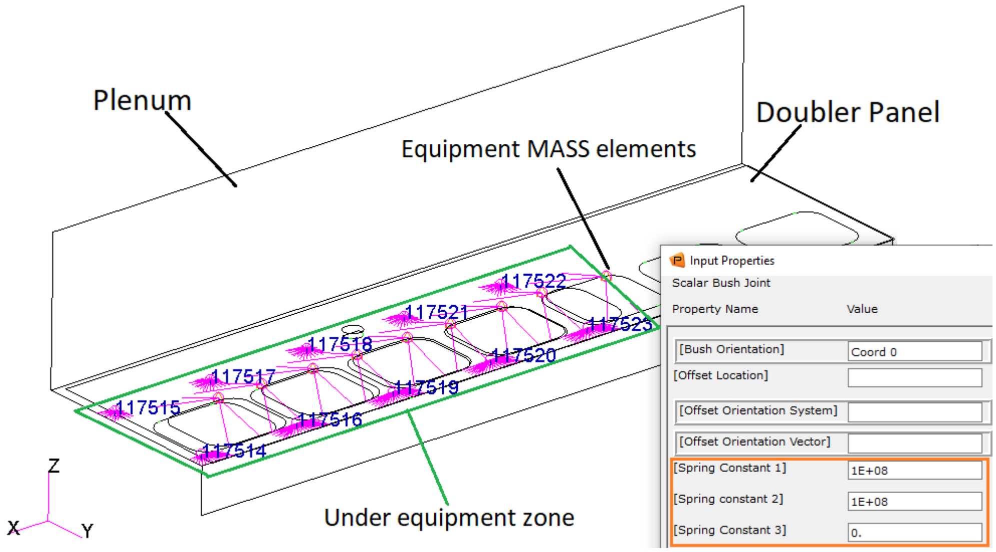


Figure 8.5. Plenum and Doubler Panel connection for “Down Only” load case set

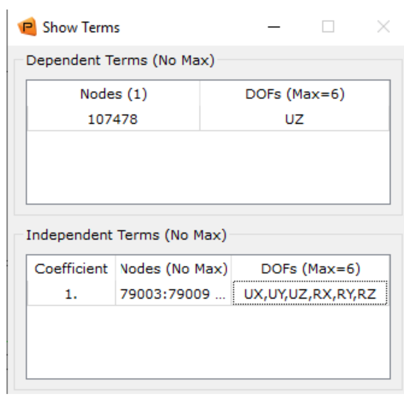


Figure 8.6. RBE3 Elements Terms for Z-load Transferring

8.1 Interface points, loads and boundary conditions

The CEC installed on Cross beams between Stow Bin rails. It fastened to Cross beams at points 1 to 6 and to the Stow Bin rail at points 7 to 14 via Shear ties. (Figure 8.7).

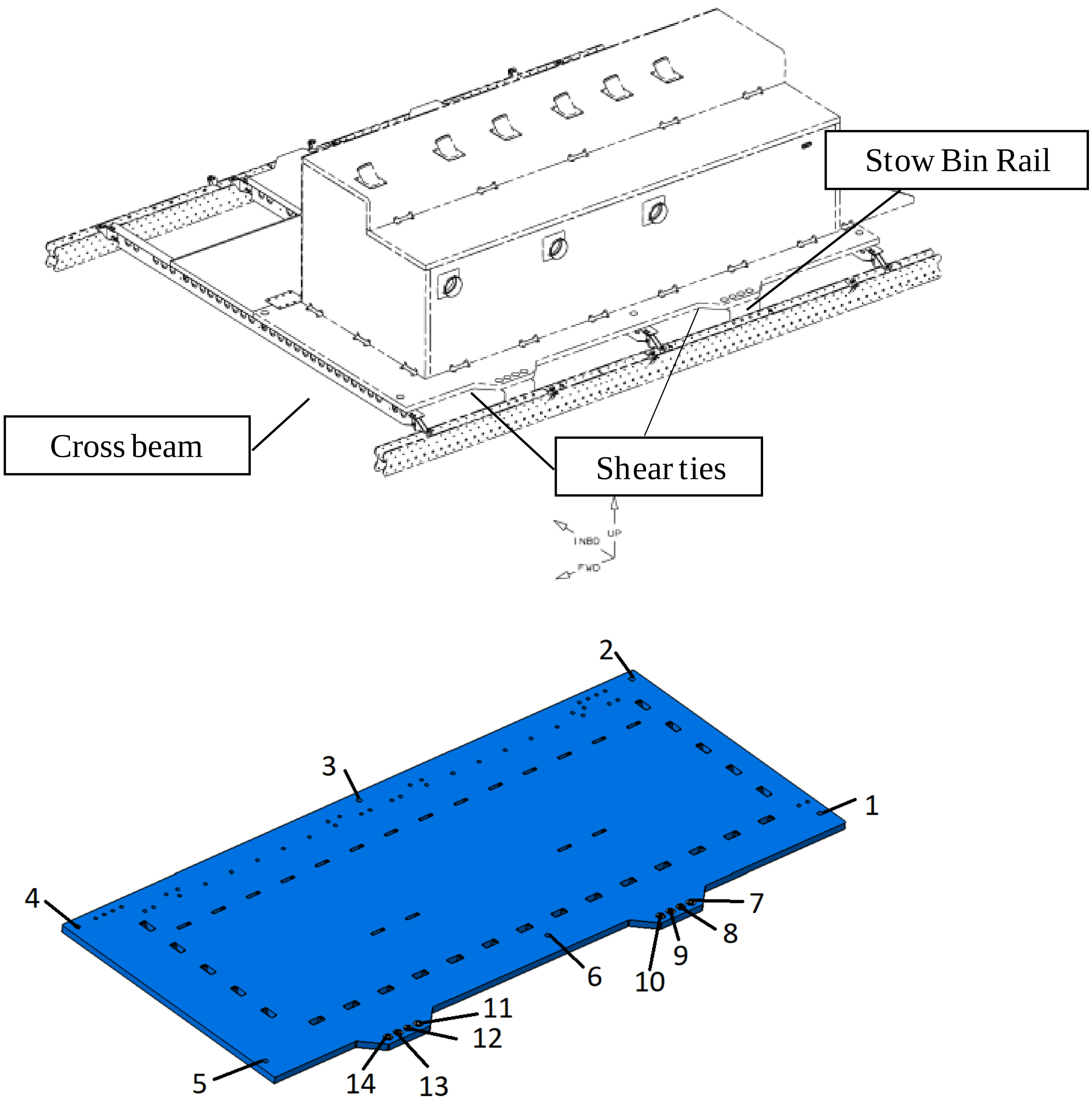


Figure 8.7. CEC Interface Fasteners Location

It is conservatively assumed that X-loads are supported only by Stow Bin rails via Shear ties. Therefore CBUSH elements implemented at interface points which have 2 translation DOF at points 1 to 6 (no X DOF) and 3 DOF at points 7 to 14

Interface points and CBUSH elements locations are presented in Table

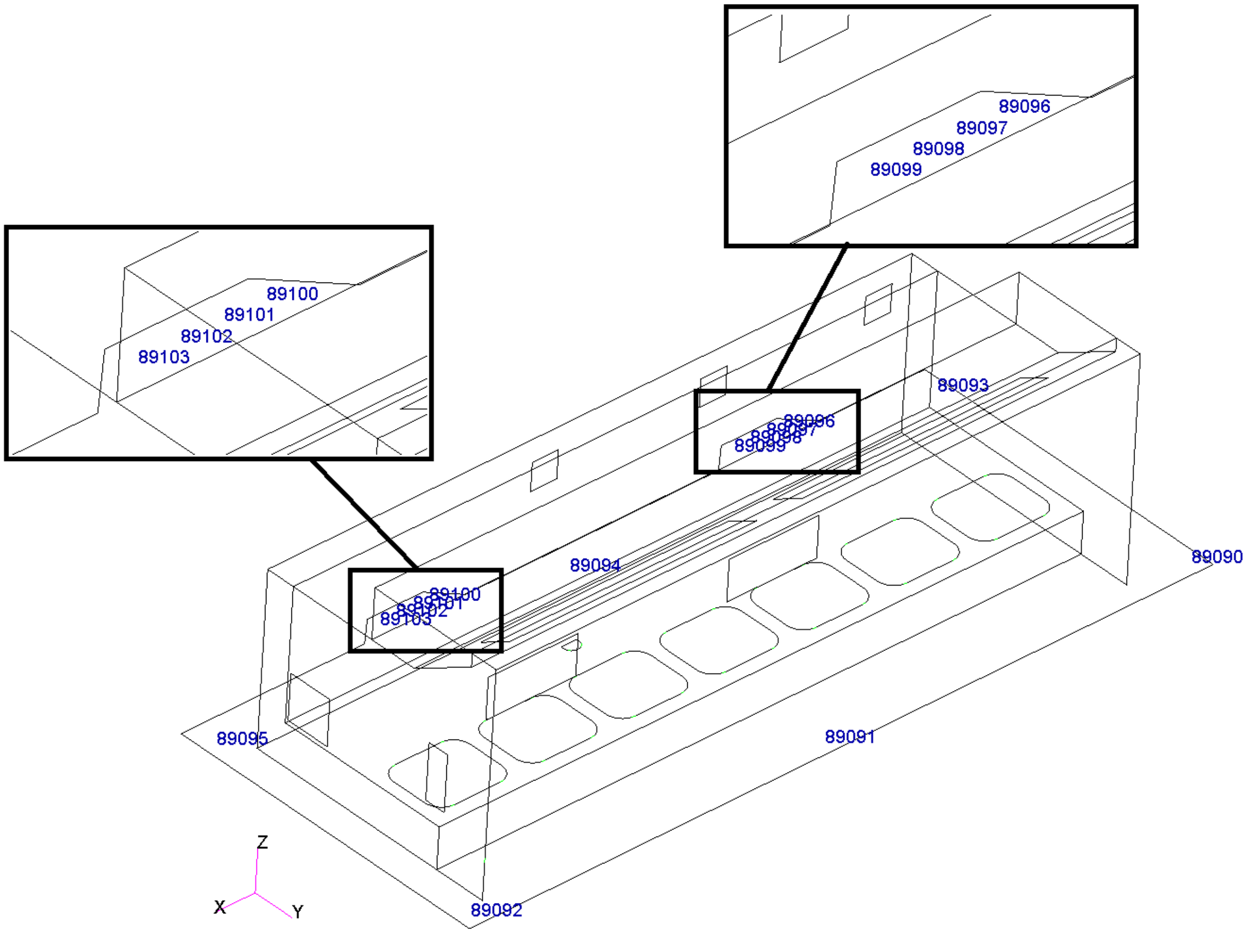


Figure 8.8. CEC Finite Element Model. Interface Element ID and Locations

Table 8.1– CEC Assy Interface Point Locations

IF point ID	Node ID	Element ID
1	95347	89095
2	95346	89092
3	95345	89091
4	95344	89090
5	95349	89093
6	95348	89094
7	95357	89103
8	95356	89102
9	95355	89101
10	95354	89100
11	95353	89099
12	95352	89098
13	95351	89097
14	95350	89096

Boundary conditions for “No Down” load set are presented in Figure 8.9.

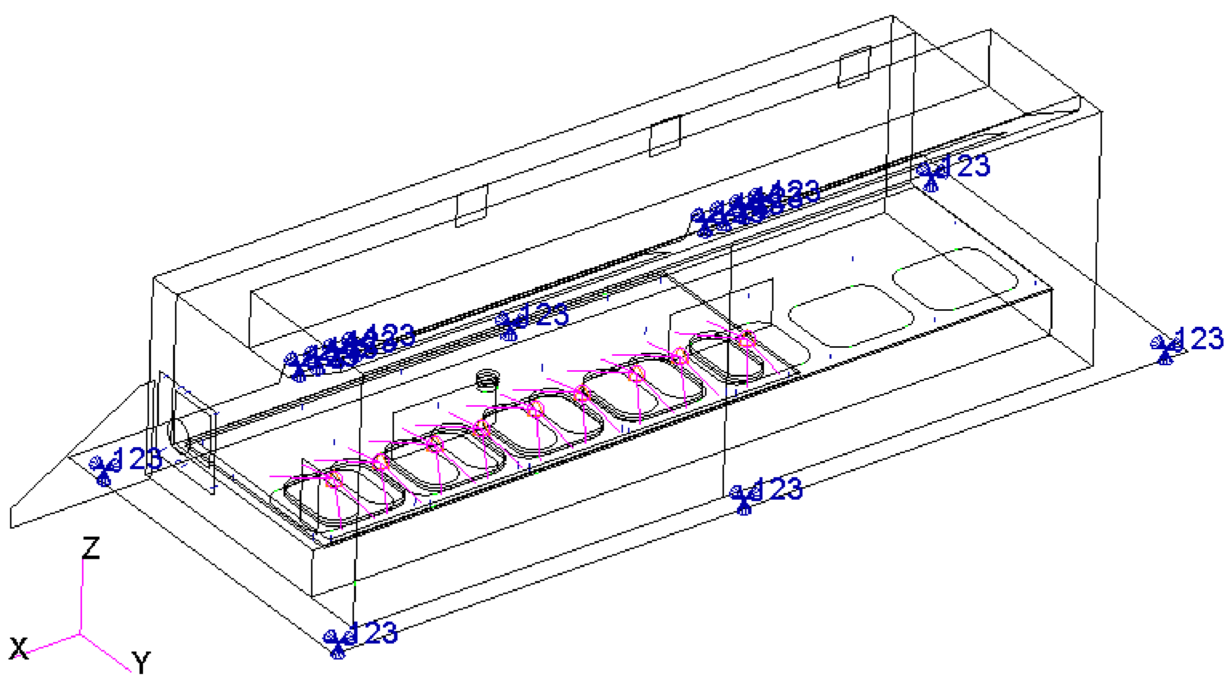


Figure 8.9 – “No Down” Load Set FEM Boundary Conditions

For “Down Only” load set additional Z-support included at the places of Cross beams.

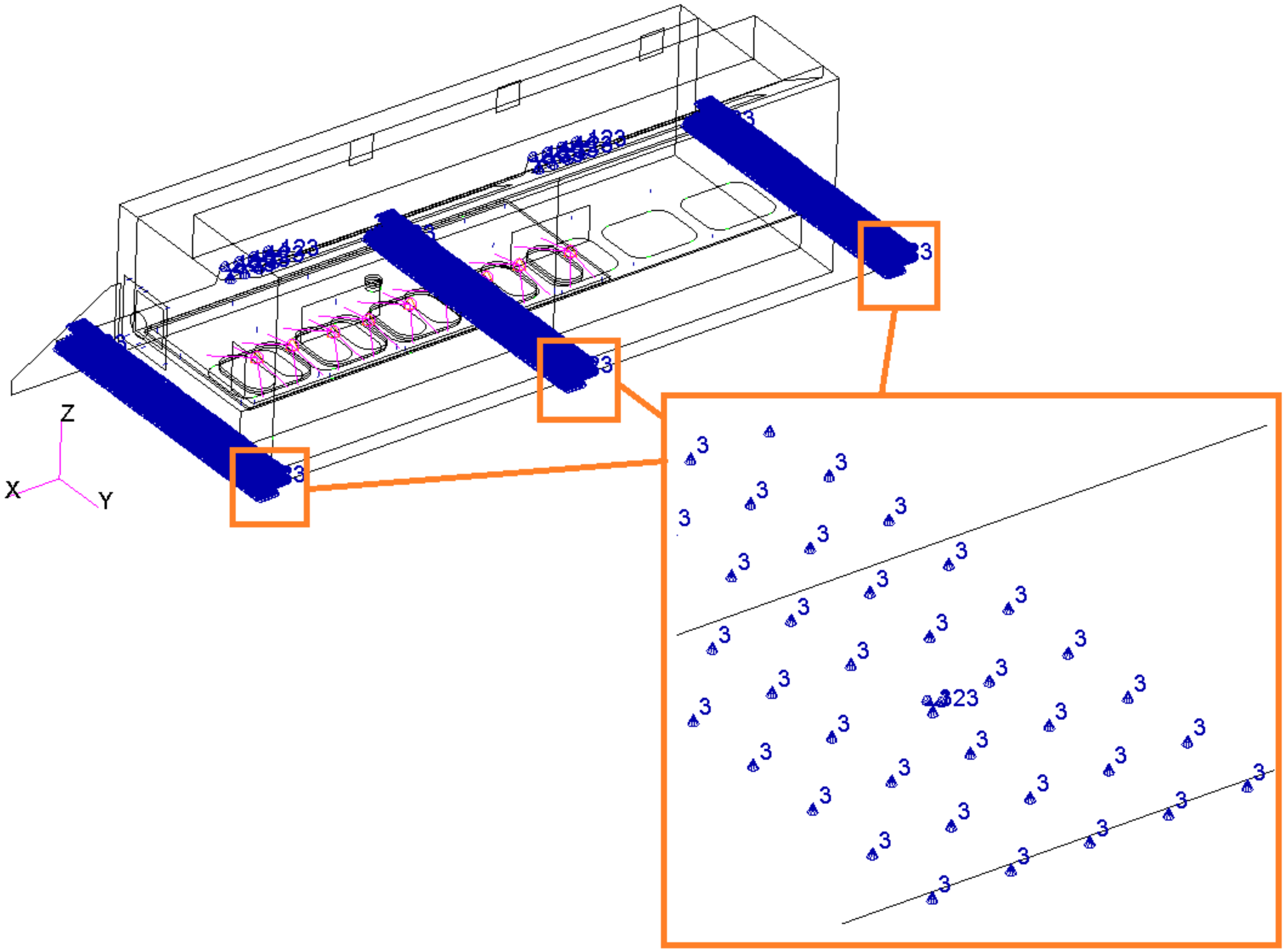


Figure 8.10 – “Down Only” Load Set FEM Boundary Conditions

8.2 Interface loads

The interface loads all tabulated below.

Table 8.2 – CEC Assy Interface Loads

Fastener #	1) 9G-Fwd			2) 1.5G-Aft			3) 3.0G-Up		
	Px	Py	Pz	Px	Py	Pz	Px	Py	Pz
1	0	402	240	0	-67	-40	0	24	147
2	0	320	111	0	-53	-19	0	-5	179
3	0	18	-46	0	-3	8	0	-8	370
4	0	-293	-77	0	49	13	0	3	69
5	0	-437	-93	0	73	15	0	14	25
6	0	-30	-48	0	5	8	0	-9	195
7	-440	197	50	73	-33	-8	-1	-2	29
8	-396	88	17	66	-15	-3	-1	-3	11
9	-402	32	6	67	-5	-1	0	-3	5
10	-457	-46	0	76	8	0	0	-5	0
11	-421	48	-39	70	-8	7	0	-3	18
12	-367	-29	-27	61	5	5	0	-2	10
13	-358	-84	-33	60	14	5	0	-1	9
14	-394	-186	-62	66	31	10	1	0	12

Fastener #	4) 6G-Dwn			5) 3G-Right			6) 3.0G-Left		
	Px	Py	Pz	Px	Py	Pz	Px	Py	Pz
1	0	-47	-294	0.0	225.2	59.0	0.0	-225.2	-59.0
2	0	10	-357	0.0	218.0	-82.0	0.0	-218.0	82.0
3	0	16	-740	0.0	46.4	-131.9	0.0	-46.4	131.9
4	0	-7	-139	0.0	85.4	-49.8	0.0	-85.4	49.8
5	0	-27	-49	0.0	87.2	12.4	0.0	-87.2	-12.4
6	0	18	-389	0.0	47.3	72.5	0.0	-47.3	-72.5
7	2	4	-58	23.2	60.9	24.9	-23.2	-60.9	-24.9
8	1	5	-23	16.0	44.2	10.8	-16.0	-44.2	-10.8
9	0	7	-11	10.7	38.2	8.6	-10.7	-38.2	-8.6
10	0	9	1	7.7	40.9	14.2	-7.7	-40.9	-14.2
11	0	6	-36	-10.0	46.2	21.6	10.0	-46.2	-21.6
12	0	4	-20	-11.6	39.6	10.8	11.6	-39.6	-10.8
13	-1	2	-18	-15.3	42.9	10.4	15.3	-42.9	-10.4
14	-1	0	-24	-20.6	56.4	18.6	20.6	-56.4	-18.6

Fastener #	7) 1.6G_Right+ 1.5G_Dwn			8) 1.6G-Left+1.5G-Dwn			9) 1.5G-Up+0.8G-Fwd		
	Px	Py	Pz	Px	Py	Pz	Px	Py	Pz
1	0	108	-42	0	-132	-105	0	48	95
2	0	119	-133	0	-114	-46	0	26	99
3	0	29	-255	0	-21	-115	0	-2	181
4	0	44	-61	0	-47	-8	0	-24	28
5	0	40	-6	0	-53	-19	0	-32	4
6	0	30	-59	0	-21	-136	0	-7	93
7	13	34	-1	-12	-31	-28	-40	16	19
8	9	25	0	-8	-22	-11	-35	7	7
9	6	22	2	-6	-19	-7	-36	1	3
10	4	24	8	-4	-19	-7	-40	-6	0
11	-5	26	2	5	-23	-21	-37	3	6
12	-6	22	1	6	-20	-11	-33	-3	2
13	-8	23	1	8	-22	-10	-32	-8	2
14	-11	30	4	11	-30	-16	-35	-17	1

Fastener #	10) 1.5G-Up+ 0.5G-Aft			11) 3.8G-Dwn+1.5G-Fwd			12) 5.6G-Dwn+0.8G-Fwd		
	Px	Py	Pz	Px	Py	Pz	Px	Py	Pz
1	0	-10	60	0	37	-146	0	-9	-253
2	0	-20	83	0	60	-208	0	38	-324
3	0	-5	187	0	13	-476	0	16	-694
4	0	18	39	0	-53	-101	0	-32	-136
5	0	31	17	0	-90	-47	0	-64	-54
6	0	-3	100	0	7	-255	0	14	-368
7	24	-12	12	-72	35	-28	-37	21	-50
8	22	-6	5	-65	18	-12	-34	13	-20
9	22	-3	2	-67	9	-6	-36	9	-9
10	25	0	0	-76	-2	1	-41	5	1
11	23	-4	11	-70	12	-30	-37	10	-37
12	21	1	6	-62	-3	-17	-33	1	-21
13	20	4	6	-60	-13	-17	-33	-6	-20
14	22	10	9	-66	-31	-26	-36	-16	-28

Fastener #	13) 6.2G-Dwn+ 0.5G-Fwd			14) 6.2G-Dwn+ 0.5G-Aft			15) 3G-Dwn+ 1.5G-Fwd		
	Px	Py	Pz	Px	Py	Pz	Px	Py	Pz
1	0	-27	-291	0	-71	-318	0	43	-107
2	0	29	-363	0	-7	-375	0	59	-160
3	0	17	-767	0	15	-762	0	11	-378
4	0	-23	-148	0	9	-139	0	-52	-82
5	0	-52	-56	0	-4	-46	0	-86	-40
6	0	17	-405	0	20	-400	0	4	-203
7	-23	15	-57	26	-7	-63	-72	35	-21
8	-21	10	-23	23	0	-24	-66	17	-9
9	-22	9	-11	23	5	-11	-67	9	-4
10	-26	7	1	25	12	1	-76	-3	0
11	-23	9	-40	23	3	-35	-70	11	-25
12	-21	2	-22	20	5	-19	-61	-3	-14
13	-21	-3	-20	19	7	-17	-60	-13	-14
14	-23	-10	-28	21	11	-22	-66	-31	-22

9 ANALYSIS RESULT

9.1 Deflections and Panel Strength

Critical Load Case for deflection is 6.2G-Dwn+0.5G-Fwd. Max deflection for this load case is 0.525 inches. This structure is not deflection critical, this is acceptable and non-linear analysis is not required.

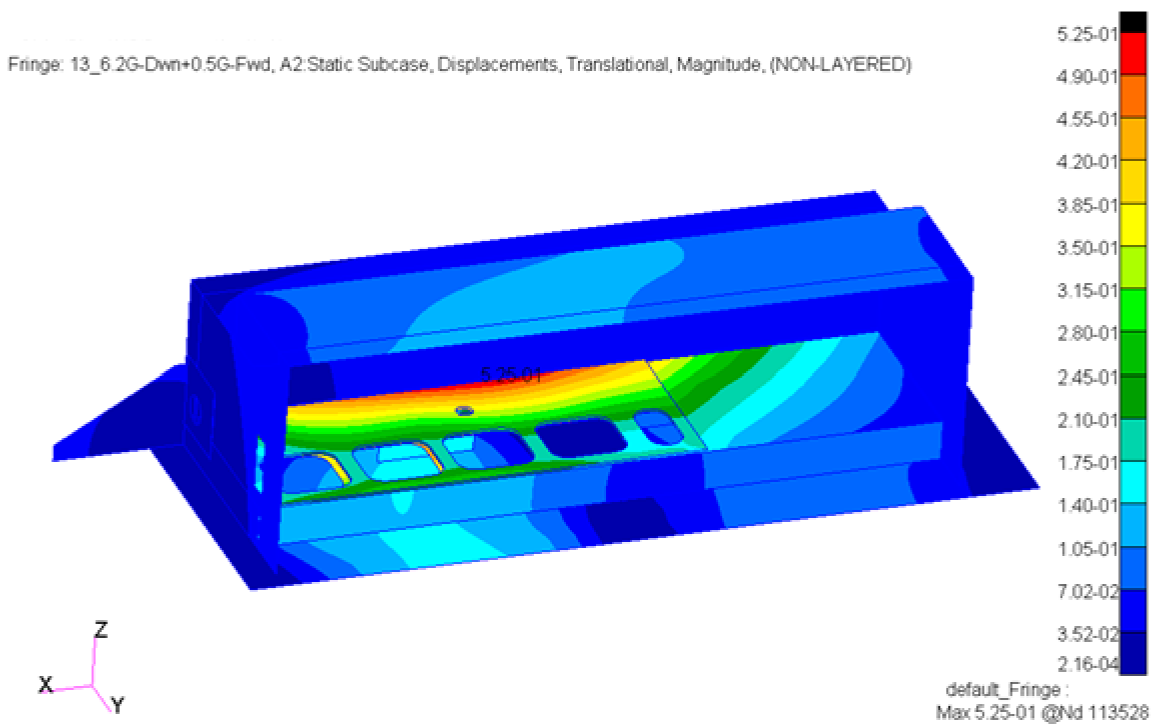


Figure 9.1. Displacement, 6.2G-Dwn+0.5G-Fwd

Figures below show the SMA margin plot for the CEC Panels assemblies. The SMA margin is a Tsai-Hill based failure criteria applicable to honeycomb structure. The areas around the Bush elements are excluded from review because these areas indicate unrealistic local pick stresses due to effect from fastener/Tab-Slot modeling specific (Bush) and/or constraints. These specific areas are commonly covered by hand analysis of attachments based on bushing internal loads extracted from FEM, but not stresses (see corresponded sections with tab and slot analysis).

Minimum Margin of Safeties are summarized in table below.

Table 9.1 – Minimum Margin of Safety Summary

Part Description	**Min. MS**	**Failure Mode**
Floor	+0.11	Core shear
Fwd Panel	+3.32	Facesheet ten/comp
Aft Panel	+2.29	Facesheet ten/comp
Back Panel	+1.79	Facesheet ten/comp
Plenum Support	+2.29	Facesheet ten/comp
Plenum	+0.23	Core shear
Filter Panel	+4.11	Core shear
Ceiling	+1.44	Core shear
CEC Doubler Panel	+0.18	Core shear
Doubler Plate	ok by inspection	-
Plenum / Aft Panel	+0.79	Combined Shear and Tension
Doubler Panel Inserts	+0.02	Tension
Doubler Panel Inserts	+1.55	Combined Tension and Shear
Floor Panel Inserts	+0.18	Combined Tension and Shear
Bolt	ok by inspection	-

Floor

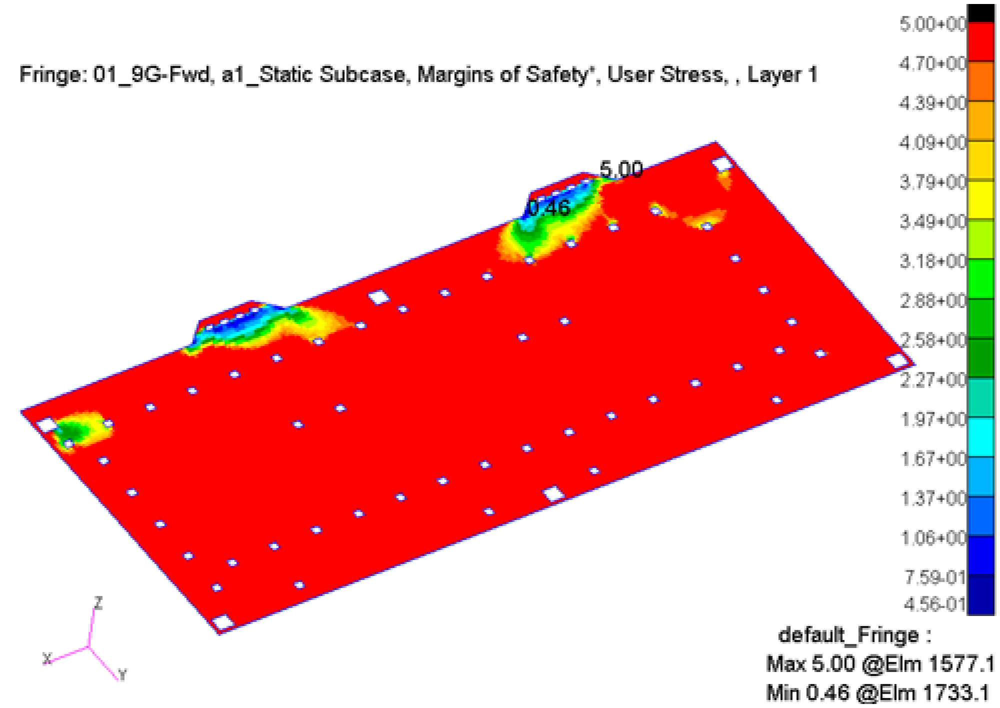


Figure 9.2. Floor Panel Facesheet Minimal MS

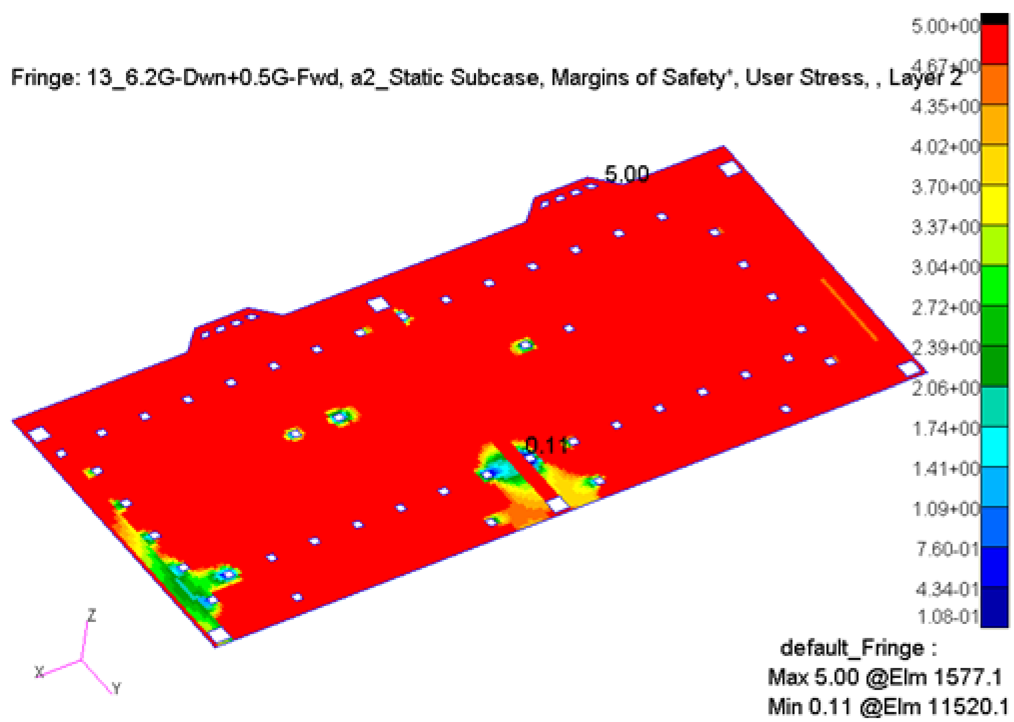


Figure 9.3. Floor Panel Core Minimal MS

Fwd Panel

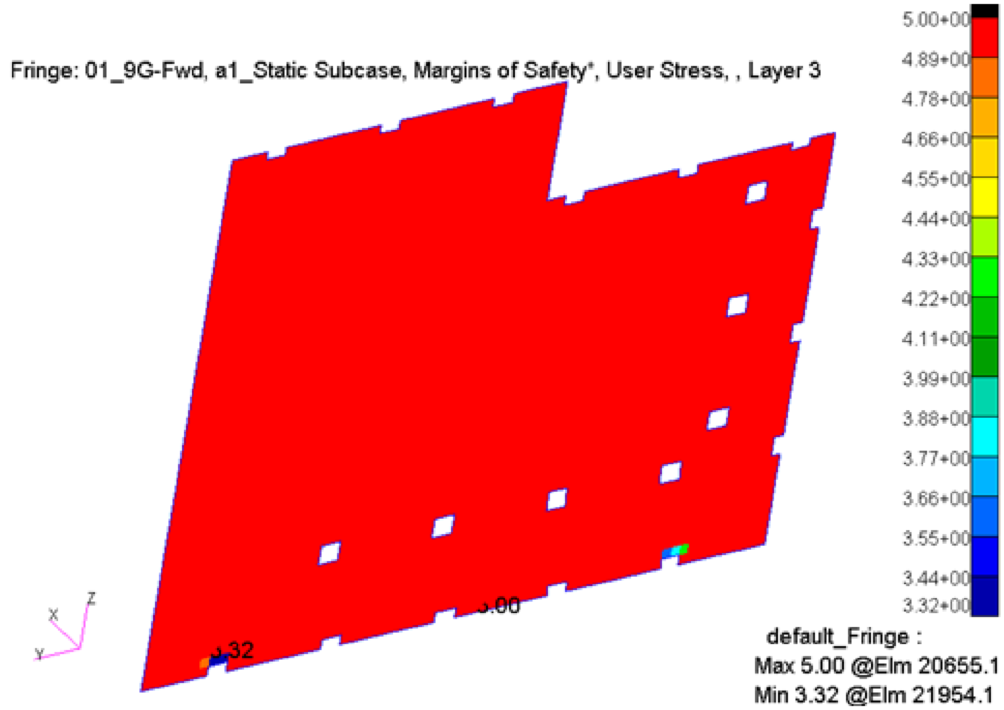


Figure 9.4. Fwd Panel Facesheet Minimal MS

Core MS of the Fwd Panel is HIGH for all Load Cases.

9.4.1 Aft Panel

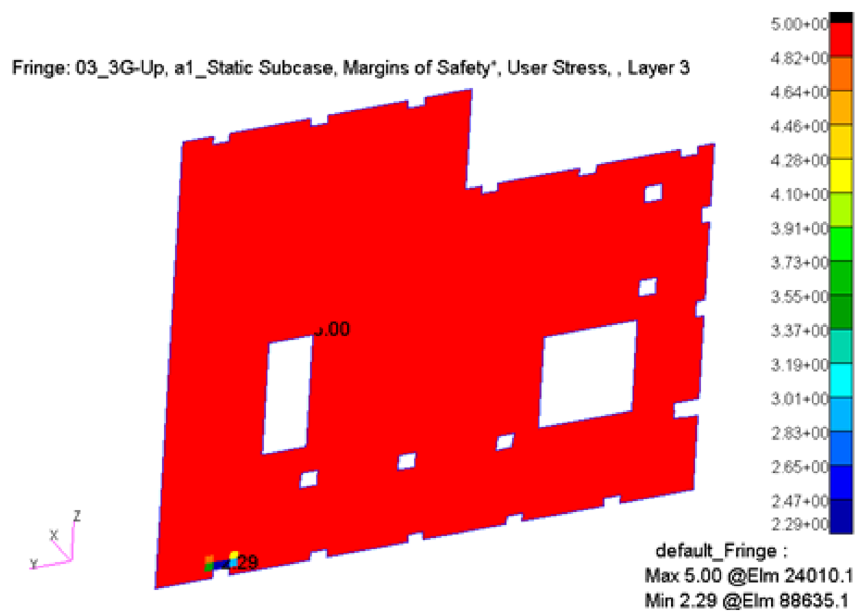


Figure 9.5. Aft Panel Facesheet Minimal MS

Core MS of the Aft Panel is HIGH for all Load Cases.

Back Panel

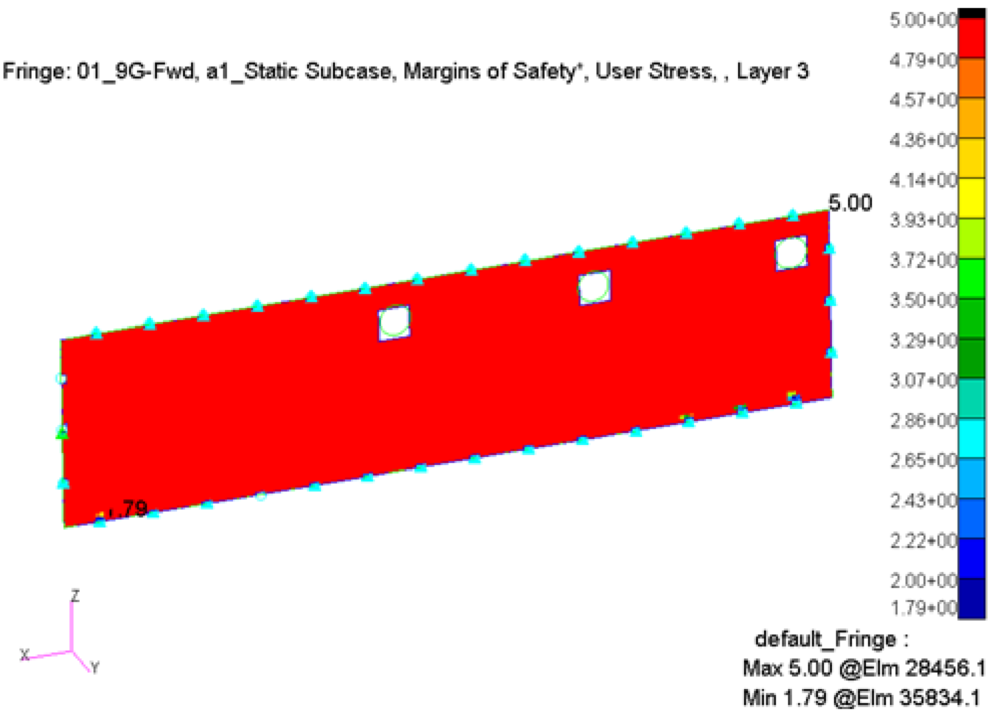


Figure 9.6. Back Panel Facesheet Minimal MS

Core MS of the Back Panel is HIGH for all Load Cases.

Plenum Support

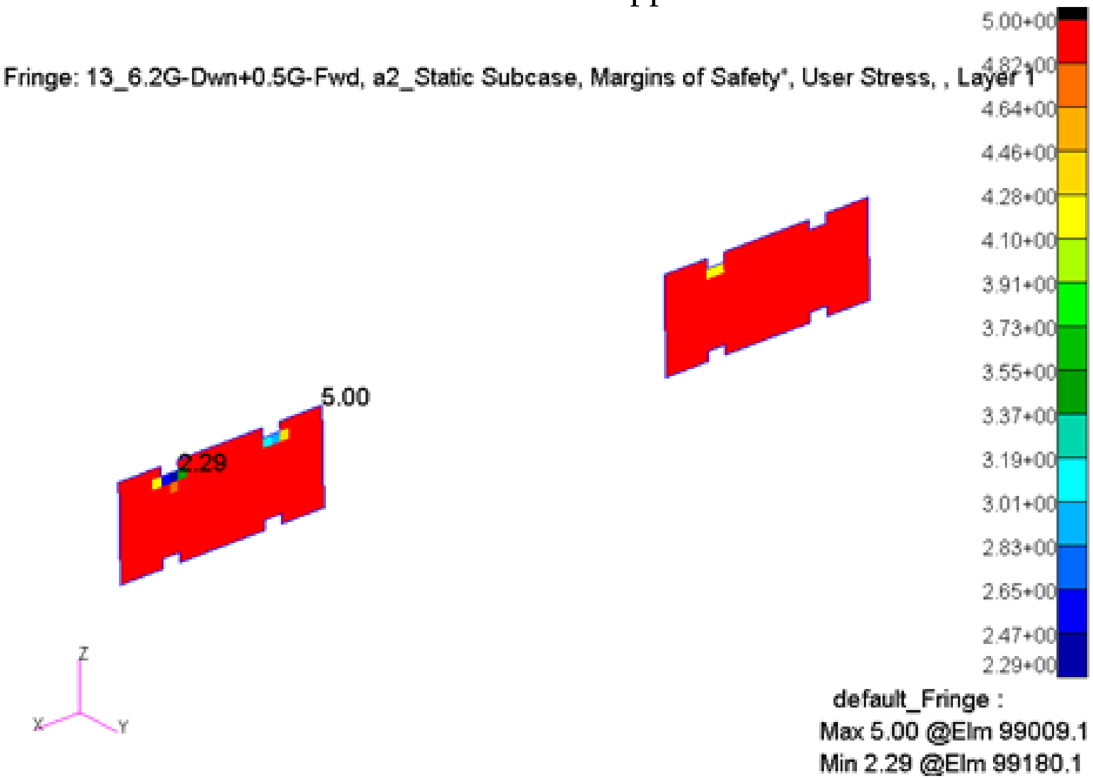


Figure 9.7. Plenum Support Facesheet Minimal MS

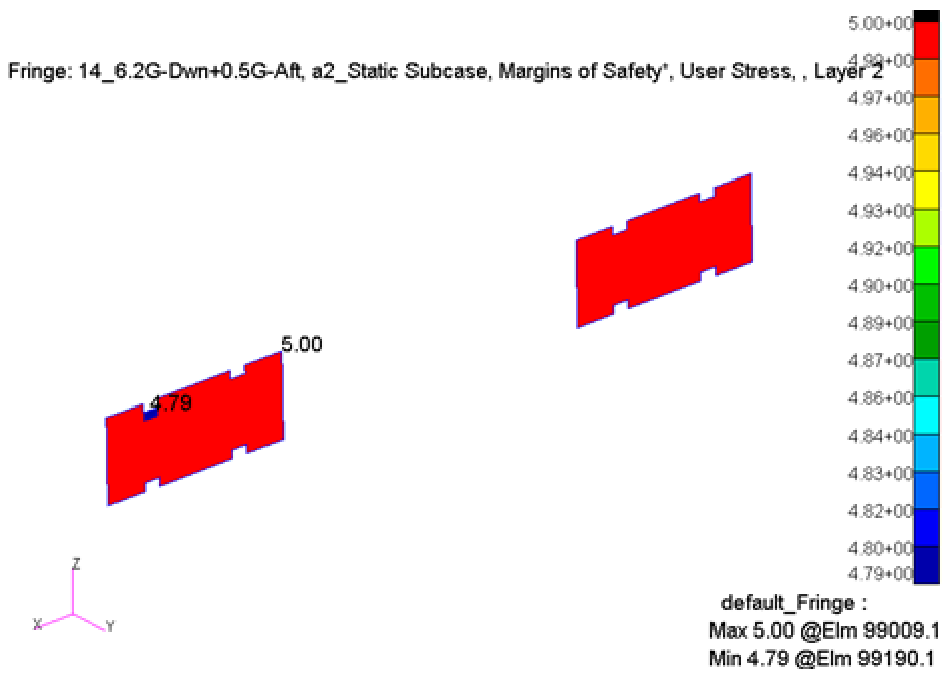


Figure 9.8. Plenum Support Core Minimal MS

Plenum

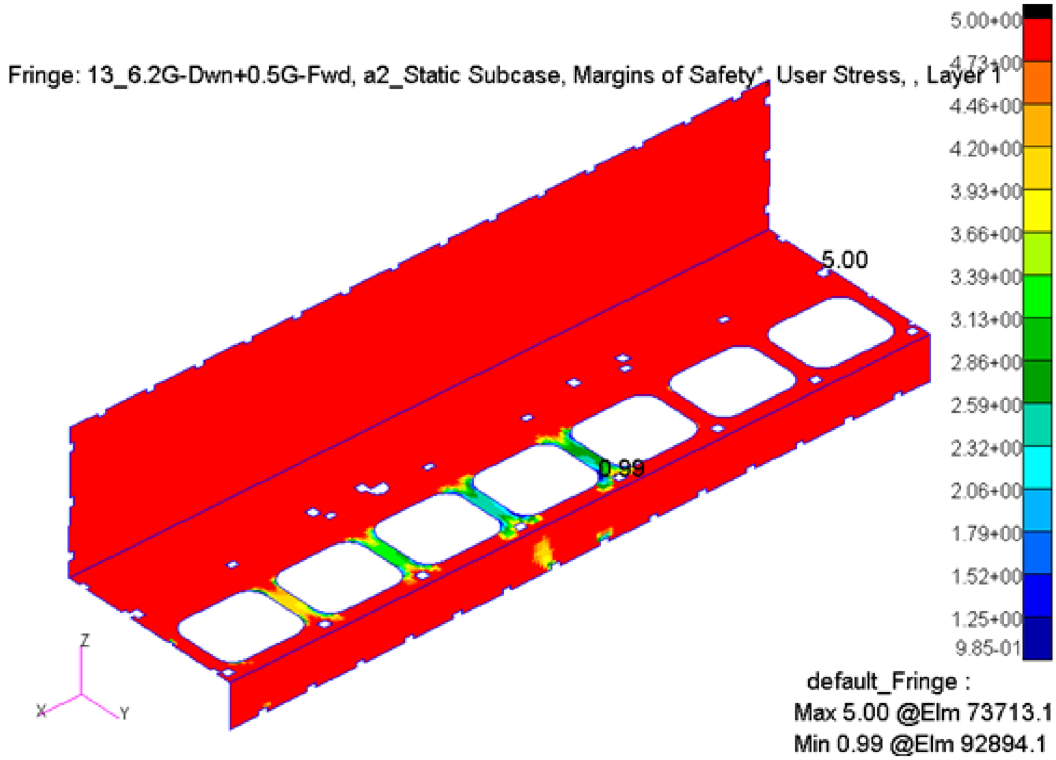


Figure 9.9. Plenum Facesheet Minimal MS

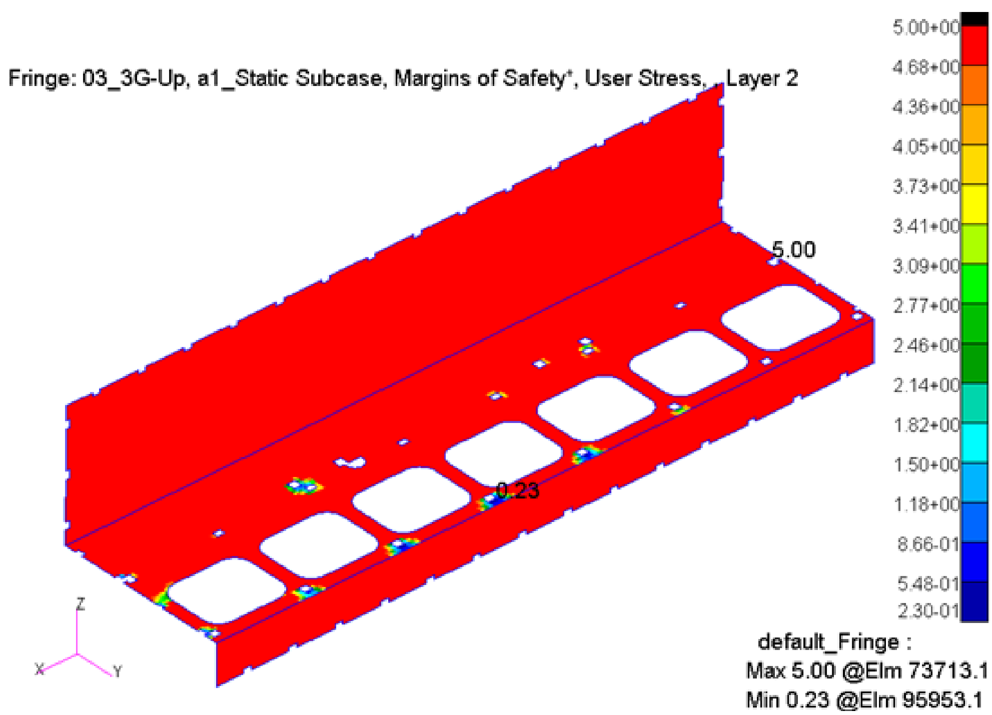


Figure 9.10. Plenum Core Minimal MS

Filter Panel

Filter Panel carries low level of internal loads at Facesheets, MS are HIGH (>+5.0) for all Load Cases and, therefore, is not critical and ok by inspection.

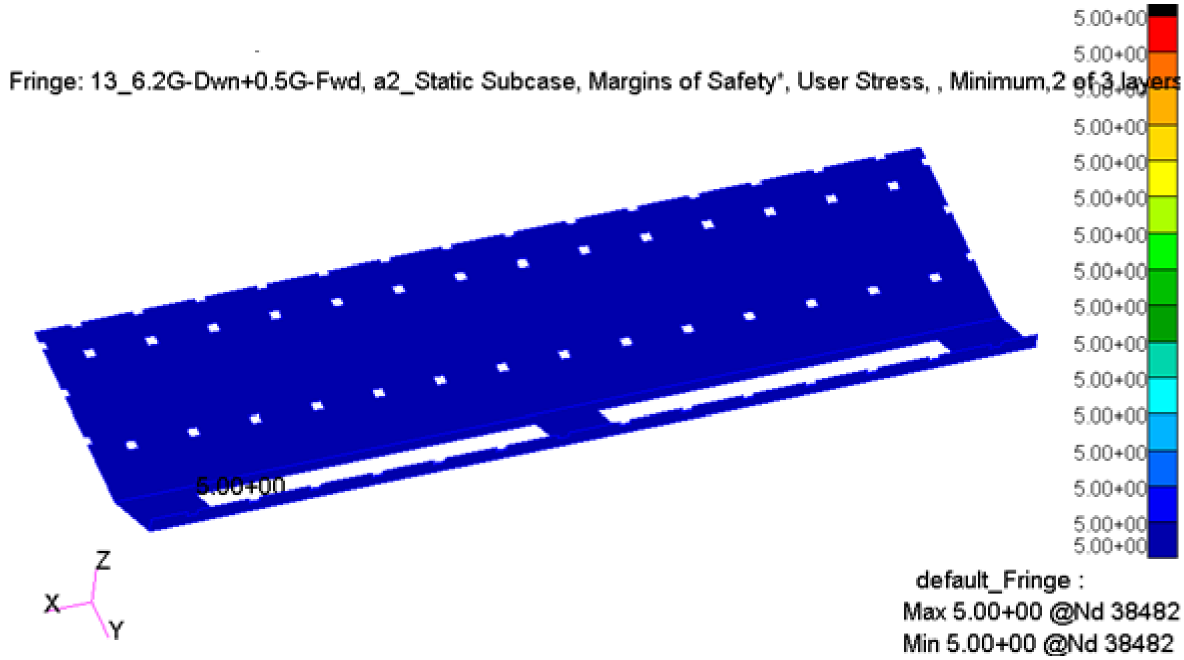


Figure 9.11. Filter Facesheet Minimal MS

Figure 9.12. Filter Core Minimal MS

Ceiling Panel

Facesheet MS of the Ceiling Panel is HIGH (> 5.0) for all Load Cases.

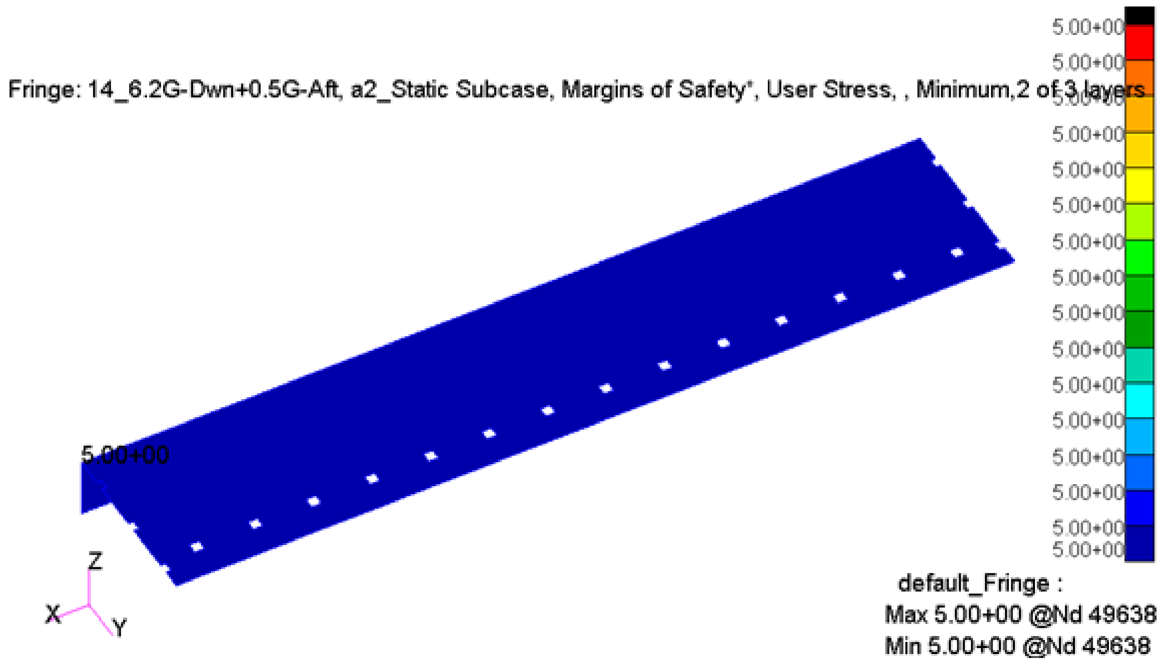


Figure 9.13. Ceiling Facesheet Minimal MS

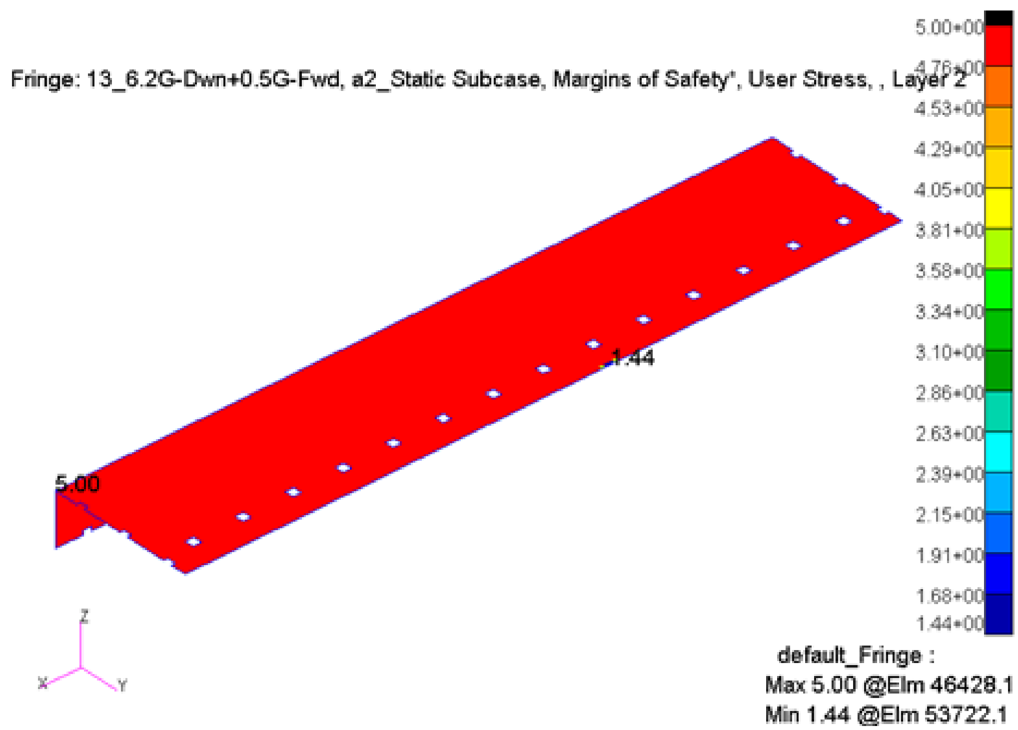


Figure 9.14. Ceiling Panel Core Minimal MS

CEC Doubler Panel

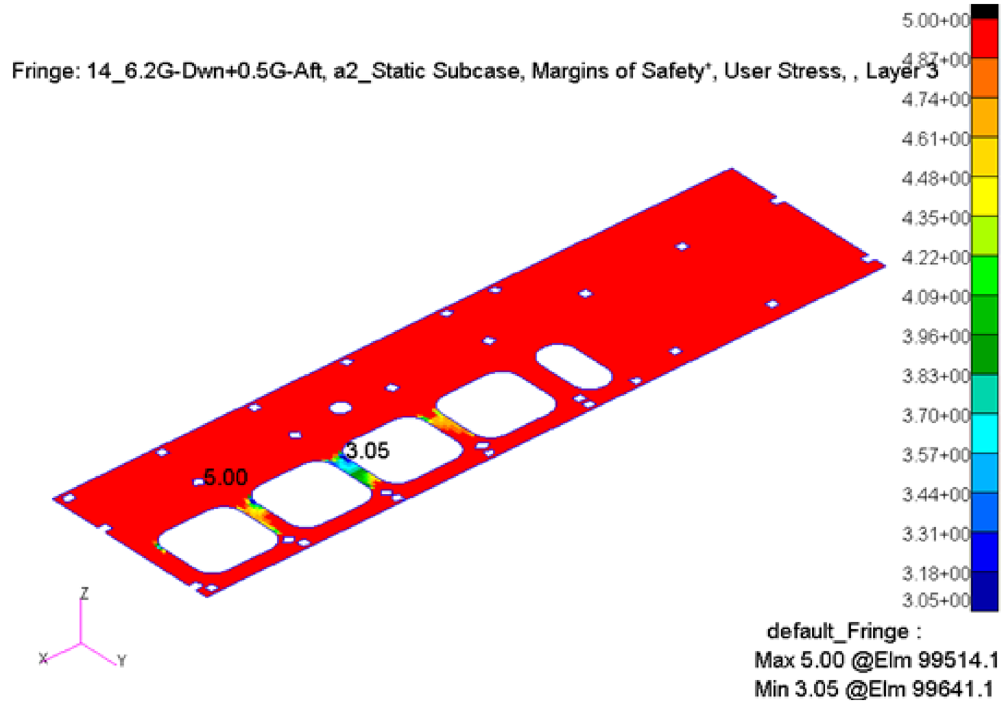


Figure 9.15. Doubler Panel Facesheet Minimal MS

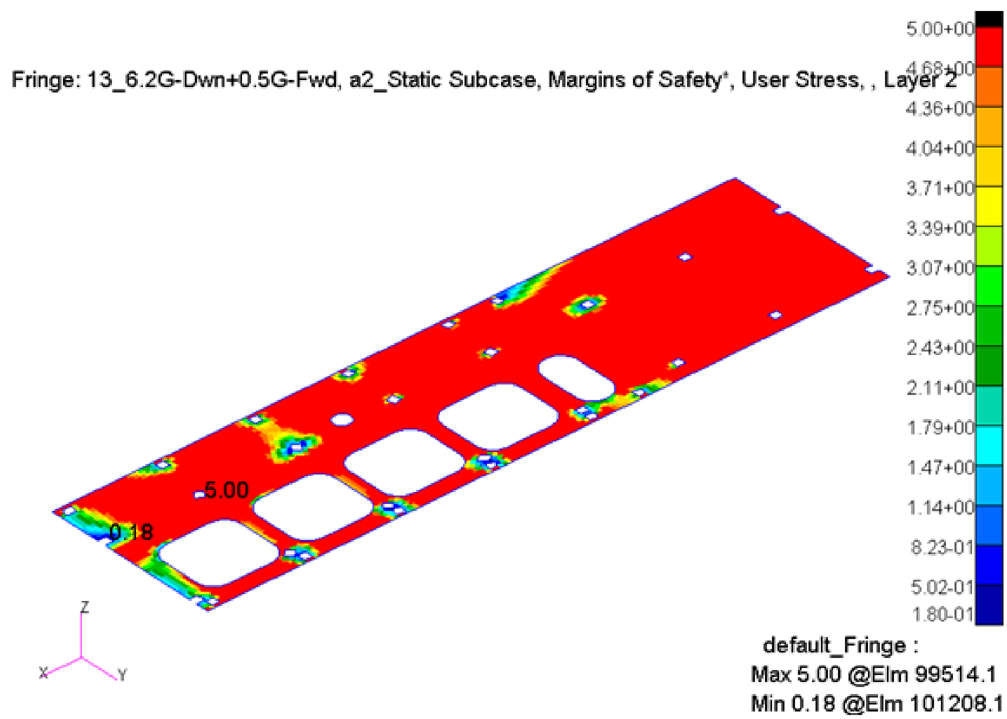


Figure 9.16. Doubler Panel Core Minimal MS

9.2 Tab and Slot Joints Strength

Tab and Slot (TS) Joint designations are presented in Fig. 9.17.

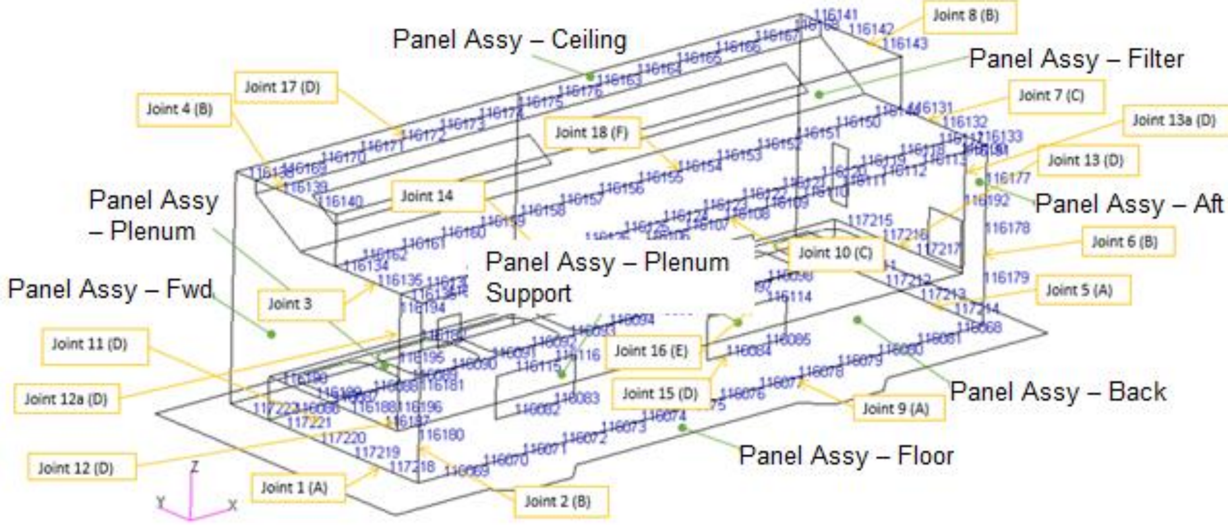


Figure 9.17. CEC Tan and Slot Joints Designations

All TS-joints are grouped by type. TS allowable values are presented in Table 9.2.

Table 9.2 – CEC Panels Joint Configuration

Joint#	Joint Type	Part 1 (Tab)	Thick(in)	Part 2 (Slot)	Slot ED	Thik(in)
1	A	Panel Assy – Fwd	1	Panel Assy – Floor	Long	1
2	B	Panel Assy – Fwd	1	Panel Assy – Back	Short	1
3	C	Panel Assy – Fwd	1	Panel Assy – Filter	Short	0.5
4	B	Panel Assy – Fwd	1	Panel Assy – Ceiling	Short	1
5	A	Panel Assy – Aft	1	Panel Assy – Floor	Long	1
6	B	Panel Assy – Aft	1	Panel Assy – Back	Short	1
7	C	Panel Assy – Aft	1	Panel Assy – Filter	Short	0.5
8	B	Panel Assy – Aft	1	Panel Assy – Ceiling	Short	1
9	A	Panel Assy – Ceiling	1	Panel Assy – Floor	Long	1
10	C	Panel Assy – Ceiling	1	Panel Assy – Filter	Short	0.5
11	D	Panel Assy – Filter	0.5	Panel Assy – Floor	Long	1
12	D	Panel Assy – Filter	0.5	Panel Assy – Fwd	Long	1
13	D	Panel Assy – Filter	0.5	Panel Assy – Fwd	Long	1

Continue of Table 9.2						
14	D	Panel Assy – Filter	0.5	Panel Assy – Aft	Long	1
15	D	Panel Assy – Filter	0.5	Panel Assy – Aft	Long	1
16	E	Panel Assy – Filter	0.5	Panel Assy – Floor	Long	0.5
17	D	Panel Assy – Plenum Support	0.5	Panel Assy – Floor	Long	1
18	E	Panel Assy – Plenum Support	0.5	Panel Assy – Plenum	Long	0.5

The strength of the tab and slot joints is checked by the SmartBush Bushing Tool per IRC standard tab and slot analysis methodology. Figure 9.18 shows the tab and slot margin plot from SmartBush tool for 14. 6.2G-Dwn+0.5G-Aft load case.

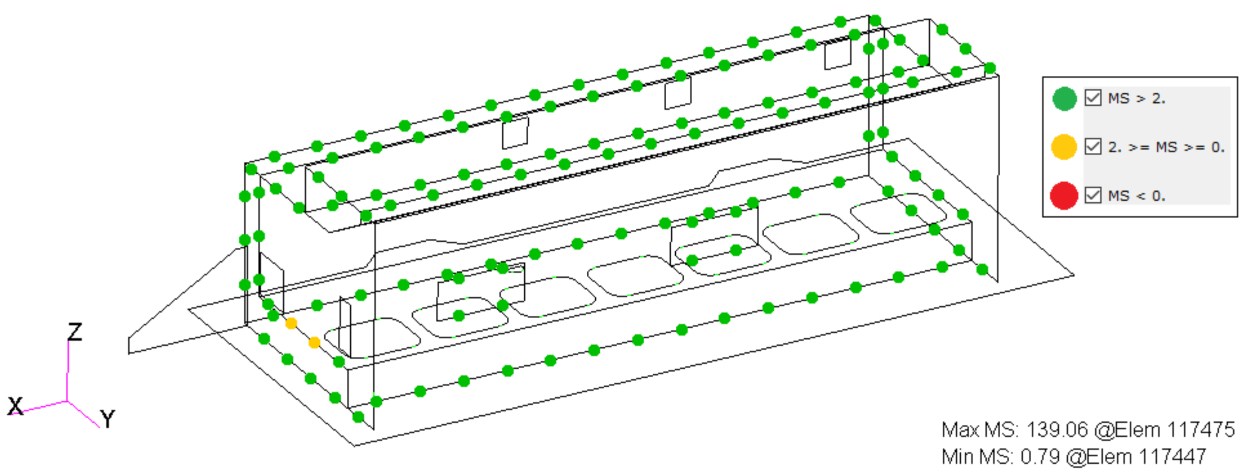


Figure 9.18. CEC Assy Minimal Tab and Slot Joint Margin of Safety

There is no any TS-joint in red, therefore all MS are positive.

9.3 Insert Strength

Inserts are analyzed per IRC methodology presented on IRC on-line resource. Detailed analysis of Inserts is presented with Margin of Safety summary, presented in Table 9.3.

Table 1 – Insert Minimum Margin of Safety

Identification	Insert Type	Load Case	Node/Elem ID	Failure Mode	Min MS	Joint Analysis
Plenum (No DWN)	A3	03_3G-Up	105586	Combined Tension & Shear	0.29	1
Plenum (DWN Only)	A3	03_6.2G-Dwn+ 0.5G-Fwd	117520	Combined Tension & Shear	0.78	2
Doubler	NAS	03_6.2G-Dwn+ 0.5G-Fwd	107481	Combined Tension & Shear	1.55	3
Floor	16C	Case:01_9.0G-Fwd	89095	Combined Tension & Shear	0.18	4

Notes: No 1.15 factor shall be used for several inserts in a row since that is a continuous joint, provided that the strength of these joints has been proven by limit and ultimate load test in which actual stress conditions are simulated in these joints and the surrounding structures.

Example of Floor panel insert analysis is presented below for "9G-Fwd" load case:

Insert Type –16C;

Tension Design Value Ft=760 lbs

Applied Tension Load Pt=240 lbs

Shear Design Value Fs=760 lbs

Applied Shear Load Ps=402 lbs

For combined loading, find the ration for applied tension and shear to their Design Values:

RT=Pt/Ft: In this case RT=240/760=0.32;

RS=Ps/Fs: In this case RS=402/760=0.53;

Margin of Safety:

$$MS = \frac{1}{R_T + R_S} - 1 = \frac{1}{0.32 + 0.53} - 1 = +0.18$$

Maximum applied values are for vibrational load cases:

VIB 16Gz up (Point 5)	Applied Tension Load Pt=507.2 lbs
VIB 16Gy right (Point 6)	Applied Shear Load Ps=497.6 lbs

Insert Design Values of A3 installed in Doubler Panel:

Tension Design Value Ft=516 lbs (Table 6.6);

Shear Design Value Fs=1193 lbs (Table , ED=0.5).

There is no combination between tension and shear loads.

Margin of Safety for tension:

$$MS = \frac{516}{507.2} - 1 = 0.02$$

Margin of Safety for shear:

$$MS = \frac{1193}{497.6} - 1 = 1.4$$

9.4 Ditch and Pot Analysis

Ditch and Pot (DAP) applications are used for analysis of a flat panel is folded to achieve an angled construction. The DAP fold is created by routing to remove a channel of facesheet and core leaving only one continuous facesheet. The Plenum and Ceiling panels folded at 90-degree angle and filled with potting compound (Joints 1, 2 and 5). The Filter panel has two folds at 120 and 150 degree angle (Joints 3 and 4). The only structural purpose of these folds is to transfer in-plane shear loads in tension from one side of the fold to the other and to add stiffness to the assembly. The fold is capable of carrying a small moment through the adhesive but cannot be relied on for structural capability and ignored in analysis.

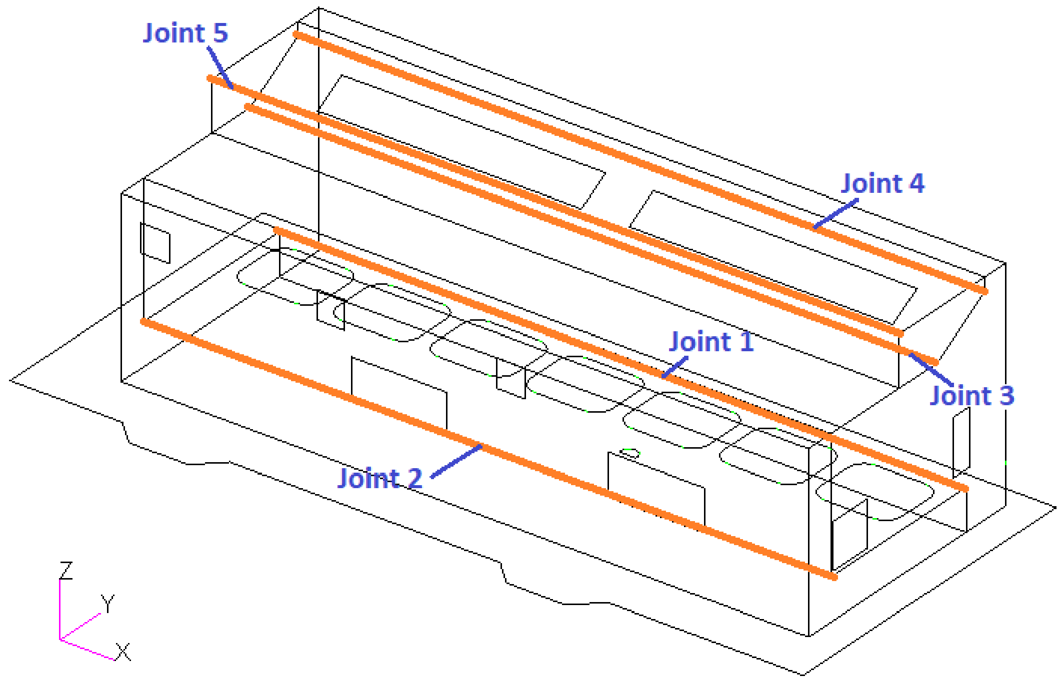


Figure 9.19. CEC Assy Ditch&Pot Joints

The min MS of the DAP in the most critical cases are shown below.

Table 9.4 –Minimum Margin of Safety

Joint#	Panel Name	DAP Type	Load Case	Node/Elem ID	ailure Mode	Min Ms
1	Plenum	Internal	13_6.2G-Dwn+ 0.5G-Fwd	115268	Coreshear	0.10
2	Plenum	Internal	14_6.2G-Dwn+ 0.5G-Aft	115537	Coreshear	3.87
3	Filter	Internal	01_9.0G-Fwd	115540	Coreshear	11.02
4	Filter	Internal	13_6.2G-Dwn+ 0.5G-Fwd	115792	Coreshear	7.22
5	Ceiling	Internal	14_6.2G-Dwn+ 0.5G-Aft	115886	Coreshear	2.17

9.5 Metallic Parts Strength Analysis

Doubler Plate does not carry any significant load. No stress analysis required and the part is ok by inspection.

9.6 Fastener Strength Analysis

Envelop fasteners/inserts loads shown in Table 9.5.

Table 9.5 – Fastener Envelope Loads

Fastener #	**Direction**	**Force, lbs**	**Load Case**
105593	Max Shear	214	9G FWD
105586	Max Tension	85	3G UP

Maximum applied tension loads is 85l lbs.

Minimum tension allowable for the fasteners is 2600 lbs.

Maximum applied shear load for the fasteners in subject is 214 lbs.

Minimum shear allowable for the fasteners in subject is 2690 lbs.

Therefore, all fasteners are considered “Good by Inspection” due to low applied loads comparing to their allowables.

10 STARTUP PROJECT DEVELOPMENT

10.1 Description of the project idea

The section analyzes the marketing analysis of a startup project, identifies opportunities and feasibility of its introduction to the market.

Table 10.1 Description of the startup project

Project content	Areas of application	Benefits for users
Determination of the stress-strain state of a composite structure	Mechanical engineering industry, Aircraft design	1) Accurate assessment 2) Speed and quality of results 3) Possibility to adapt the considered model to different cases of loading

The proposed technique allows determining the required level of strength for composite construction at any load conditions in a short time and with sufficient accuracy.

10.2. Technology audit

It is possible to realize the idea of the project through field tests and statistical analysis. In the Table 10.2 the analysis of potential technical and economic advantages of this idea in comparison with the competitor # 1 (foreign colleagues in the field of aircraft and mechanical engineering industry).

Table 10.2 Determination of strong, weak and neutral characteristics of the project idea

№	Technical and economic characteristics of the idea	W	N	S
1	Cash	Competitor №1	-	My project
2	Method of assessment	-	Competitor №1	My project
3	Complexity of calculation	-	-	-

Table 10.3 Technological feasibility of the project idea

№	The idea of the project	Technology of its implementation	The presence of technology	Technology availability
1	Determination of the stress-strain state of a composite structure	Simple interface	yes	yes
		Quick access in different devices		
The selected technology can be implemented.				

According to the indicators of the state of the market, we can conclude that this project is profitable.

10.3. Analysis of market opportunities for launching a startup project

Determining the market opportunities that can be used in the market implementation of the project, and market threats that may impede the implementation of the project, is quite difficult, given that different methods of solving the task is an element of long-term scientific development of the industry. That is, to evaluate the potential market for a startup project is possible only in the long run, not based on clear numerical characteristics of the market. Let's analyze the market opportunities for the implementation of our project. To begin with, we will conduct a demand analysis: demand availability, volume and dynamics of market development Table 10.4.

Table 10.4 Preliminary description of a potential startup project market

№	Market state indicators	Characteristics
1	Number of main players, units	2
2	Total sales, UAH / unit	100
3	Market dynamics	increase
4	Sign-in restrictions	Absent
5	Specific requirements for standardization and certification	available
6	Average rate of return in the industry,%	100%

According to the indicators of the state of the market, we can conclude that this project is profitable.

Identification of potential customer groups

Potential customer groups can be roughly divided into primary and secondary customers. The primary group is the district and regional aircraft. In the future, we will identify potential customer groups per Table 10.5.

Table 10.5 Characteristics of potential clients of a startup project

№	The need that shapes the market	Target audience	Differences in behavior of different potential target customers	Consumer requirements for the product
1	Design of composite structure	Boeing subsidiaries	Finances	Speed of the determination and simplicity

Given the competitive situation, there is an opportunity to work in this market. To be competitive in the market, a project must have characteristics such as the speed of calculation and the availability of software.

Based on the analysis of competition conducted, and taking into account the characteristics of the idea of the project, consumer requirements for the table and factors of the marketing environment, determine and justify the list of factors of competitiveness. The analysis is formalized in Table 10.6.

Table 10.6 Rationale for competitiveness factors

№	Competitiveness factor	Rationale (citing factors that make the comparison of competing projects meaningful)
1	less need for costs	No need for repeat operations
2	Test accuracy	Improving results
3	The speed of calculation	Maximum resource depletion

According to the identified factors of competitiveness Table 10.6 we will analyze the strengths and weaknesses of my startup project Table 10.7.

The final stage of market analysis of project implementation opportunities is the compilation of SWOT analysis (Strength and Weak matrix, Troubles and Opportunities on the basis of selected market threats and opportunities, and strengths and weaknesses Table 10.7.

Table 10.7 Comparative analysis of strengths and weaknesses "Design of metal-composite compound structure by means of manual calculations”

№	Competitiveness factor	Points 1-20	Competitive rating of products compared to the project "Design of metal-composite compound structure by means of manual calculations "						
			-3	-2	-1	0	1	2	3
1	less need for costs	20				●			
2	Accuracy of calculations	20			●				
3	Using the data obtained	20					●		
4	The accuracy of the calculation in the project	15					●		

The list of market threats and market opportunities is compiled on the basis of an analysis of threat factors and factors of the marketing environment. Market threats and market opportunities are the effects of factors and, by contrast, have not yet been realized in the market and are likely to occur.

Based on the SWOT analysis, market behavior alternatives are developed for launching a startup project to the market and an approximate optimal timing of their market implementation in view of potential competitors' projects that may be launched. The identified alternatives are analyzed in terms of timing and likelihood of receiving resources Table 10.8.

Table 10.8 Alternatives to market introduction of a startup project

№	An alternative to market behavior	The probability of receiving resources	Terms of implementation
1	Public review, review of existing studies (analogues), state approval	high	3 months
2	Publication, validation of the present experiment, state approval	high	10 month

From the above alternatives, we will choose the first one, because obtaining resources is simpler and more likely and the timing of implementation is shorter.

11 CONCLUSION

Cabin Entertainment Center (CEC) Installations are structurally acceptable by analysis. The analysis results have shown that all Margins of Safety are positive, which means that the safety conditions are met.

Therefore, the CEC 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 CEC can be installed in an aircraft by attaching it only to ceiling beams of an aircraft. The developed finite element model can be used to calculate the strength of other similar structures.

REFERENCES

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

3. Michael C.Y. Niu. Composite Airframe structures practical design information and data, 1992.

4. Michael C.Y. Niu. Atrframe stress analysis and syzing, 2nd ed, jan., 1999.

5. Lee Young-Hwan, Do-Wan Lim, Jin-Ho Choi, Jin-Hwe Kweon and MyungKeun Yoon, Composite Structures, volume 92, 2010.

6. Kolesnikkov B., Herbeck L. and Fink A., Composite Structures, volume 83, 2008.

7. https://en.wikipedia.org/wiki/Finite_element_method

8. https://patents.google.com/patent/EP2040980A2

9. http://www.npfasteners.com/composites/deform-nut.htm

10.https://www.flightglobal.com/future-visions-new-in-flight-entertainment-for-the-airbus-a380-and-boeing-787-examined/69350.article

11. Airworthiness Standards FAA CFR Part 25.

12. Бланк, С. Стартап. Настольная книга основателя / С. Бланк, Б. Дорф ; пер. с англ. Т. Гутман, И. Окунькова, Е. Бакушева. – 2-е изд. – Москва : Альпина Паблишер, 2014.