

**МІНІСТЕРСТВО ОСВІТИ І НАУКИ УКРАЇНИ
НАЦІОНАЛЬНИЙ ТЕХНІЧНИЙ УНІВЕРСИТЕТ УКРАЇНИ
“КИЇВСЬКИЙ ПОЛІТЕХНІЧНИЙ ІНСТИТУТ ім. ІГОРЯ
СІКОРСЬКОГО ”**

**МЕХАНІКО-МАШИНОБУДІВНИЙ ІНСТИТУТ
ДИНАМІКА І МІЦНІСТЬ МАШИН ТА ОПОРУ МАТЕРІАЛІВ**

"На правах рукопису"
УДК _____

«До захисту допущено»
Завідувач кафедри ДММ та ОМ
_____ Пискунов С. О.
“ ” _____ 2019 р.

**МАГІСТЕРСЬКА ДИСЕРТАЦІЯ
на здобуття ступеня магістра
зі спеціальності 131 Прикладна механіка**

**на тему: «Вплив жорсткості та компонування шпангоутів фюзеляжу на
розподіл зусиль у поздовжньому стику обшивок»**

Виконала:
студентка II курсу ОКР «Магістр»,
групи МП-83мп,
Данильчук Вікторія Олегівна

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»

Напрямок 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

_____ Данильчук Вікторії Олегівні _____.

(прізвище, ім'я, по батькові)

1. Тема дисертації _____ Вплив жорсткості та компонування шпангоутів фюзеляжу на розподіл зусиль у поздовжньому стику обшивок _____.

науковий керівник дисертації _____ Сидоренко Ю.М. к.т.н., доц. _____ ,

(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 _____ повздовжній стик обшивок

4. Предмет дослідження _____ розподіл зусиль у поздовжньому стику обшивок _____.

5. Перелік завдань, які потрібно розробити дослідити вплив жорсткості та компонування шпангоутів фюзеляжу на розподіл зусиль у поздовжньому стикі обшивок.

6. Орієнтовний перелік ілюстративного матеріалу 53 рисунка

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	02.09.19-01.10.19	Виконано
2	Ознайомлення з типовими конструкціями повздовжніх стиків	01.10.19-15.10.19	Виконано
3	Вибір прототипної конструкції	15.10.19-01.11.19	Виконано
4	Створення та аналіз розрахункової моделі	01.11.19-15.11.19	Виконано
5	Обробка результатів	15.11.19-22.11.19	Виконано
6	Підготовка презентації по стану роботи	22.11.19-27.11.19	Виконано
7	Оформлення роботи	27.11.19-16.12.19	Виконано

Студент

(підпис)

Данильчук В.О.
(ініціали, прізвище)

Науковий керівник дисертації

(підпис)

Сидоренко Ю.М.
(ініціали, прізвище)

АНОТАЦІЯ

Дипломна робота містить 66 сторінок, 23 таблиці, 53 рисунка, список використаної літератури з 2 найменувань.

Актуальність роботи полягає в тому, що існує необхідність у визначеності впливу жорсткості та компонування шпангоутів фюзеляжу на розподіл зусиль у поздовжньому стику обшивок.

Метою дипломної роботи є пошук шляхів для вагової оптимізації конструкції літака за допомогою зміни конфігурації конструкції та аналіз впливу цих змін на критичні зони навантаження.

Як об'єкт дослідження був поздовжній стик обшивок фюзеляжу.

Для вирішення задачі проведено розрахунки чисельним методом за допомогою методу скінченних елементів.

Ключові слова: *поздовжній стик, розподіл зусиль, з'єднання заклепками.*

Abstract

Thesis contains 66 pages, 23 tables, 53 figures; reference list consists of 2 titles.

The urgency of the work is that there is a need for certainty as bone regeneration affects the rigidity of osteosynthesis systems.

The aim of the work is a need to determine the influence of rigidity and layout of the fuselage frames on the load distribution in the longitudinal lap splice.

The aim of the thesis is to find ways to optimize the weight of the aircraft structure by changing the configuration of the structure and to analyze the impact of these changes on the critical load areas as a longitudinal lap splice.

The object of the study is the longitudinal lap splice of fuselage skins.

To solve the problem, analysis is performed using the numerical method as the finite element method.

Key words: lap splice, load distribution, riveted joint.

CONTENTS

INTRODUCTION	7
1. EXISTING STRUCTURE OVERVIEW OF LONGITUDINAL SKIN SPLICES AND FRAMES ON FUSELAGE	8
2. FINITE ELEMENT METHOD. SUBSTANTIATION OF FINITE ELEMENTS TYPE AND SIZE.....	16
3. ANALYSIS OF THE LONGITUDIAL LAP SPLICE OF SKINS USING FINITE ELEMENT METHOD.....	23
4. OVERVIEW RESULTS	44
5. STARTUP DEVELOPMING.....	53
6. CONCLUSIONS	65
REFERENCE LIST	66

INTRODUCTION

The design of the aircraft structure is based on customer and manufacturer requirements, safety protocols, physical and economic constraints. The design process begins with the intended purpose of the aircraft. For example, commercial airliners are designed for the carriage of passengers or cargo with a large load capacity, longer range and greater fuel efficiency. In any case, the aircraft must be acceptable for such requirements as a high strength and durability with minimum weight.

The strength of the aircraft structure is a one of the most important requirement for flight safety, and reducing the mass, in turn, directly affects fuel efficiency. Therefore, to obtain an aircraft with high performance, its strength must obtained using the most effective methods.

Nowdays strength analysis approaches are based on experimental, empirical and theoretical experience and methods.

The dissertation is devoted to such elements of the aircraft design as the longitudinal joint of the fuselage casings. The first section is in the field of view of existing structures and explains their functions and design features. Section 2 deals with the justification of the size of a finite element for longitudinal lap splice of skins. The finite element results are processed in Section 3. The startup analysis is performed in Section 4.

Key words: *lap splice, load distribution, riveted joint.*

1. EXISTING STRUCTURE OVERVIEW OF LONGITUDINAL SKIN SPLICES AND FRAMES ON FUSELAGE

A fuselage of a modern aircraft is a stiffened shell commonly referred to as semi-monocoque construction. This type of construction, as show on figure 1.1 contains longitudinal elements (longerons and stringers) and transverse elements (frames and bulkheads) which are intended to carry the major portion of fuselage bending moment, maintain shape of a fuselage cross section and stabilize fuselage skin. The fuselage skin withstands the shear caused by applied external transverse and torsional forces, contributes to carrying of the bending moment and undergoes the action of excessive cabin pressure [1].

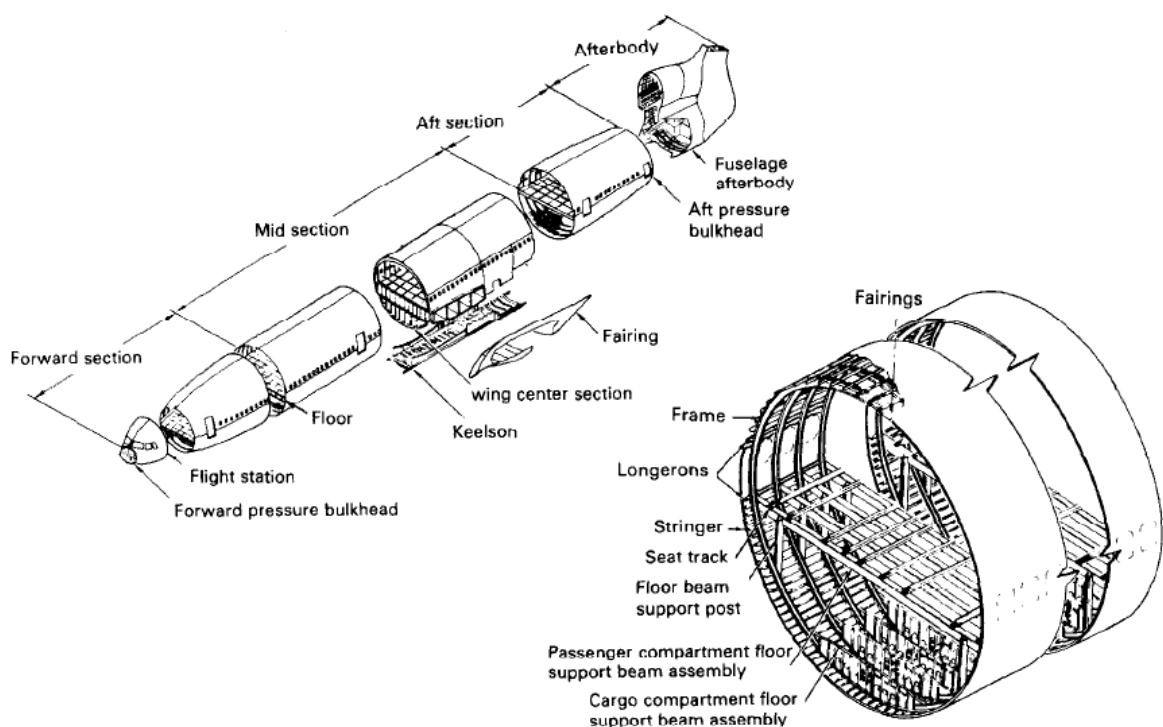


Fig. 1.1 – Typical semi-monocoque fuselage structure

In order to create comfortable environment for passengers and crew, a fuselage of an aircraft must be pressurized. The most efficient pressure-carrying structure has a cylindrical cross-section with spherical end caps, but fuselage

structure must have cutouts and openings, space for a weather radar antenna and large cavities for the wing and landing gears [1]. Figure below shows general configuration of a modern pressurized fuselage.



Fig. 1.2 – Fuselage configuration of a commercial aircraft

Figure above shows that construction of the fuselage includes pressurized and unpressurized compartments. Each cutout and unpressurized cavity must be separated by structural elements, which withstand internal cabin pressure, such as bulkheads, pressure decks and pressure webs [1].

The largest single item of the fuselage structure is the skin and its stiffeners. It is also the most critical structure since it carries all of the primary loads due to fuselage bending, shear, torsion and cabin pressure. These primary loads are carried by the fuselage skin and stiffeners with frames spaced at regular intervals to prevent buckling and maintain cross-section [1]. Fig. 1.3 below shows typical skin-stiffener panels.

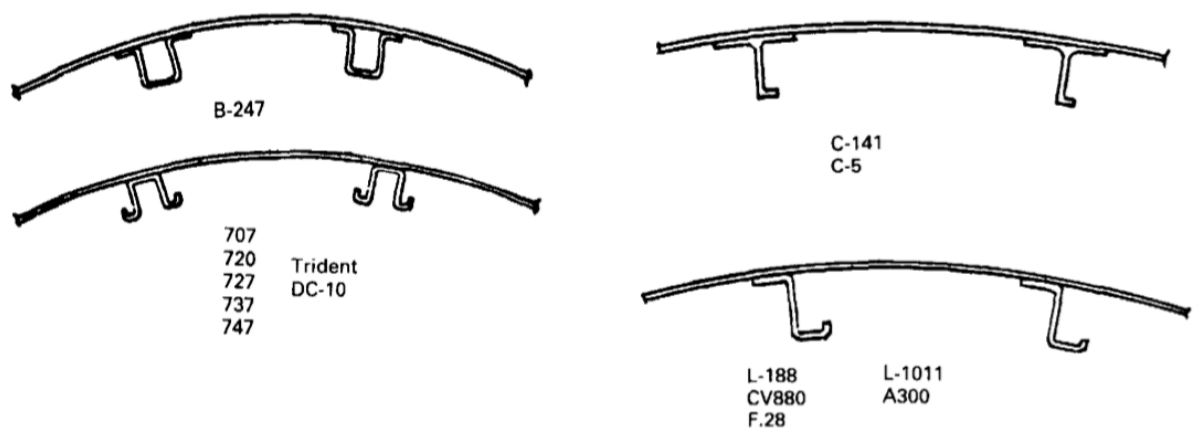


Fig. 1.3 – Typical transport skin-stringer panels

Note that there has been very little change in the basic structural concept since the earliest metal stressed skin airplanes. These skin/stiffener combinations have proved over the years to be light-weight, strong structures that are relatively easy to produce and maintain [1].

The most efficient structure is the one with the least number of joints or splices; therefore, the skin panels are as large as possible, limited only by available mill sizes. Stringers, being rolled from strip stock, are limited in length by manufacturing techniques [1].

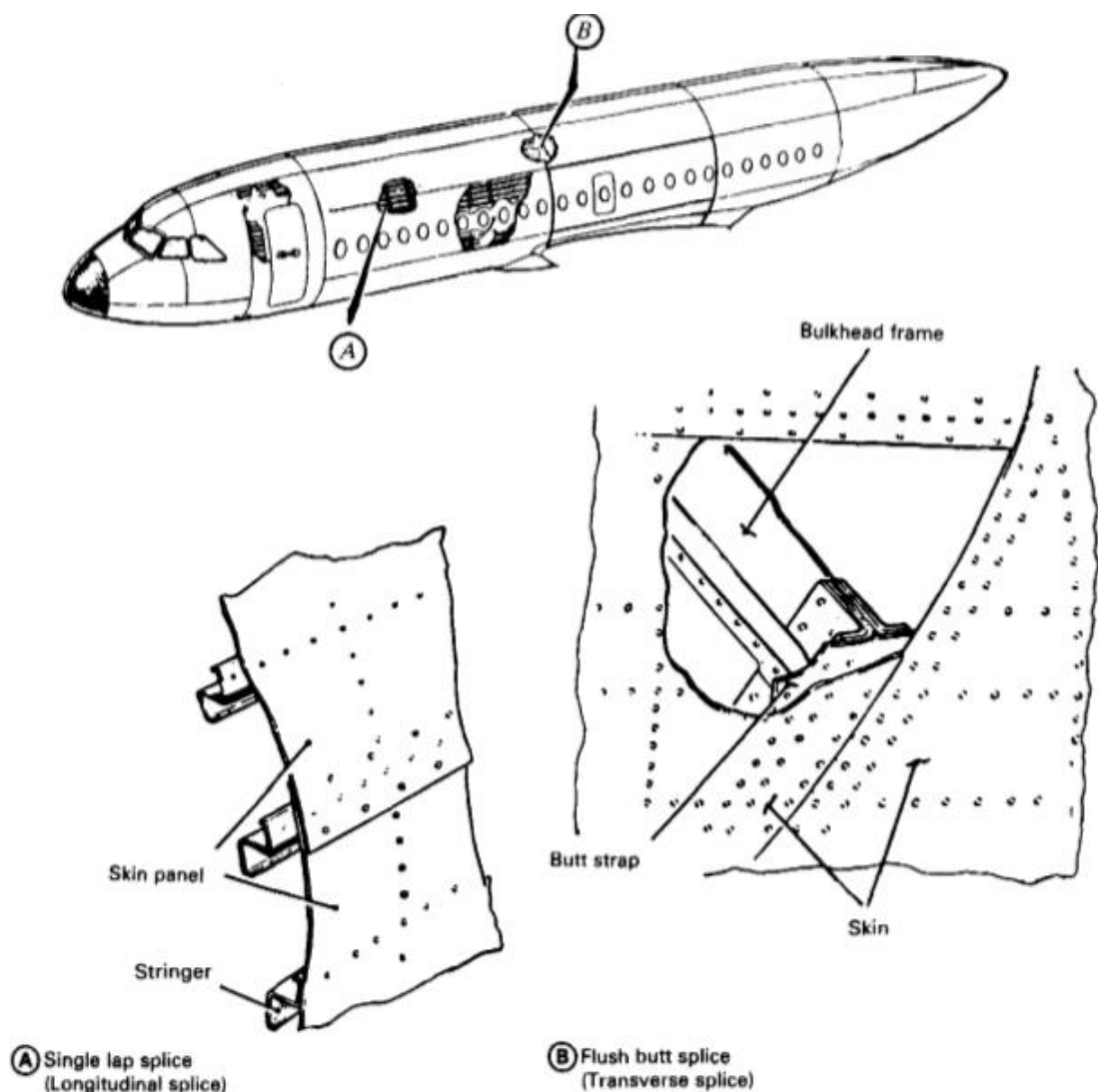


Fig. 1.4 – Typical transport fuselage splices

Single lap splices shown in Fig. 1.4 (a) are typically used for the longitudinal skin joints. This is the lightest design and does not impose a severe aerodynamic penalty on a subsonic airplane. The transverse splices, those normal to the air stream, are flush butt splices [Fig. 1.4 (b)] because a lap or step here would have an appreciable effect on boundary layer turbulence and drag [1].

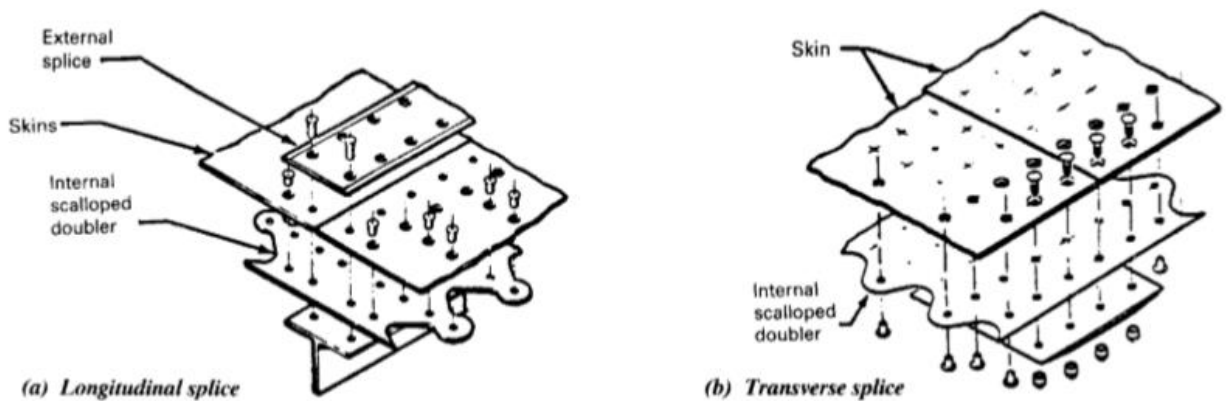


Fig. 1.5 – Fuselage splices with scalloped doublers

Fig. 1.5 illustrates the design of a typical longitudinal and circumferential splice. The circumferential splice consists of a strap doubler with a scalloped doubler placed between the strap and the skin break [1].

Stringer splice locations are established by another set of rules. Since the skin and stringers are working together, they should both be spliced at the same location. This maintains the relative stiffness of the skin/stringer combination, which is desirable from a fatigue standpoint [1].

Examination of the various typical fuselage configurations of commercial airplanes reveal that they are basically similar combinations of skin-stringer (frame) structures, with the interior trim line (one inch greater than the frame depth) providing an overall cabin wall thickness. Ring or frame spacing is in the order of 20 inches and stringer spacing varies between 6 to 10 inches. Commonly, the passenger transport fuselage sidewall (window and door area) design replaces stringers with heavier thickness skins so that a quieter cabin can be obtained and the

skin fatigue stress can be reduced because of cabin pressurization cycles. In the sidewall region, the frame depth can be kept to a minimum (provided that adequate working space is not a problem) because no significant concentrated loads are involved. Above and below the side-wall region, ample space is available to profile for increased frame depth as required. Another advantage is to reduce fuselage diameter to save structural weight and less fuselage frontal area to reduce aero-dynamic drag with the same internal width across the cabin (see Fig. 1.6) [1].

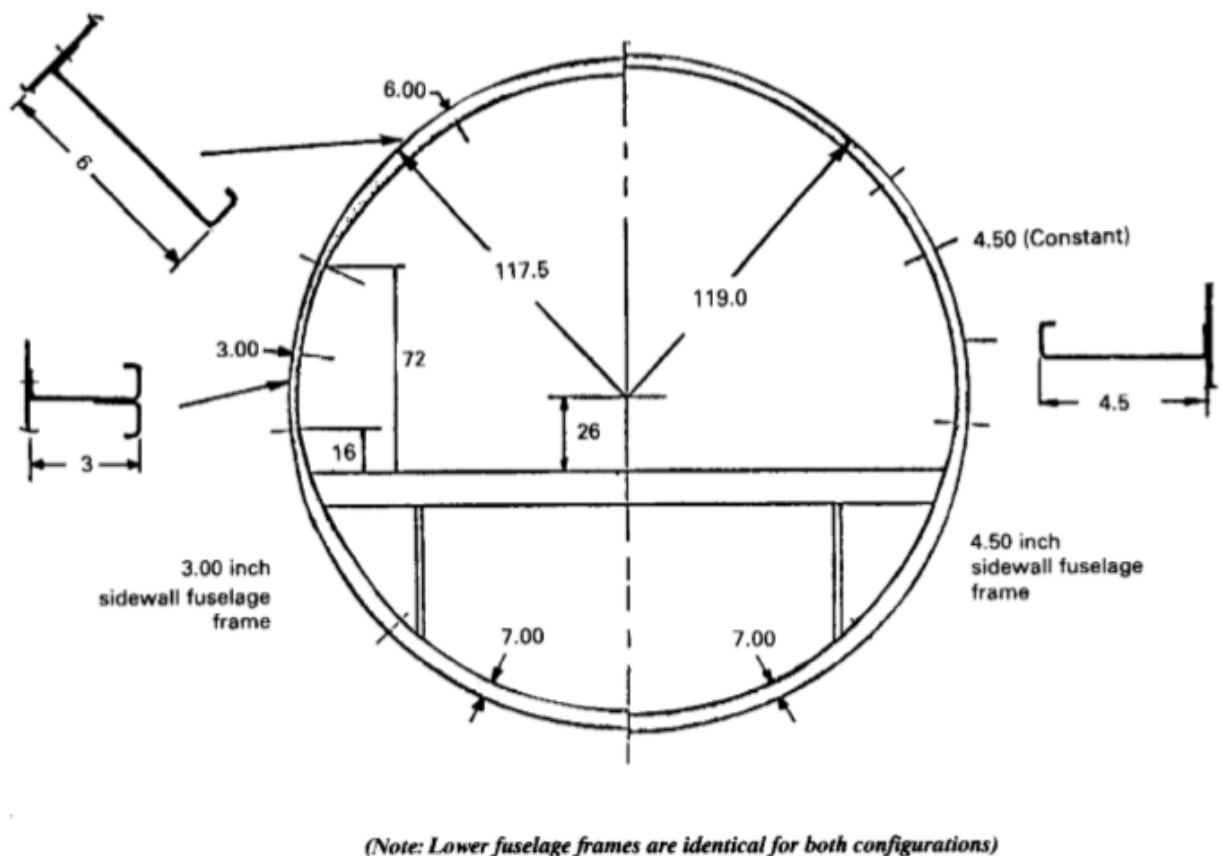


Fig. 1.6 – Effect of sidewall frame depth study of L-1011 fuselage diameter

Fig. 1.7-1. illustrate several examples of trans-port fuselage skin, stringer and frame arrangements; see below [1].

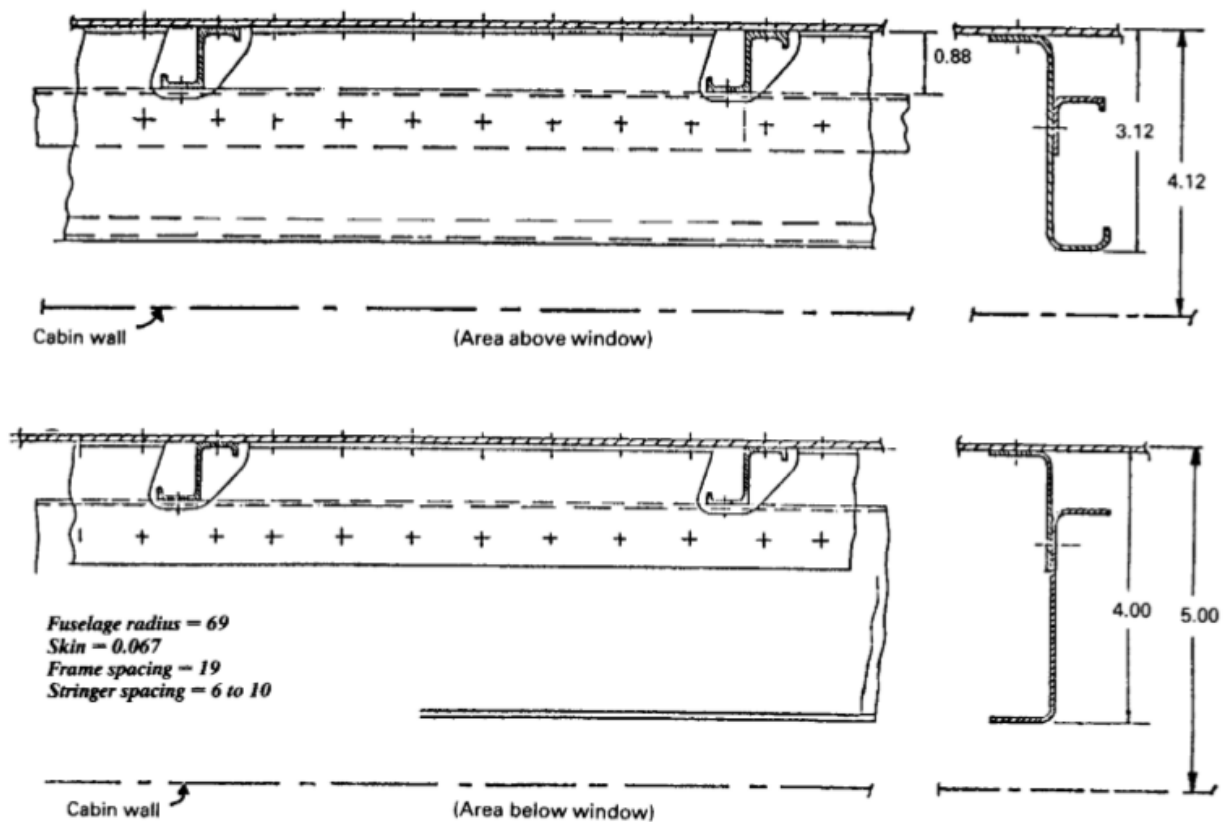


Fig. 1.7 – Convair 880. Transport fuselage structural arrangement. (Dimensions in inch and materials are aluminum except as noted)

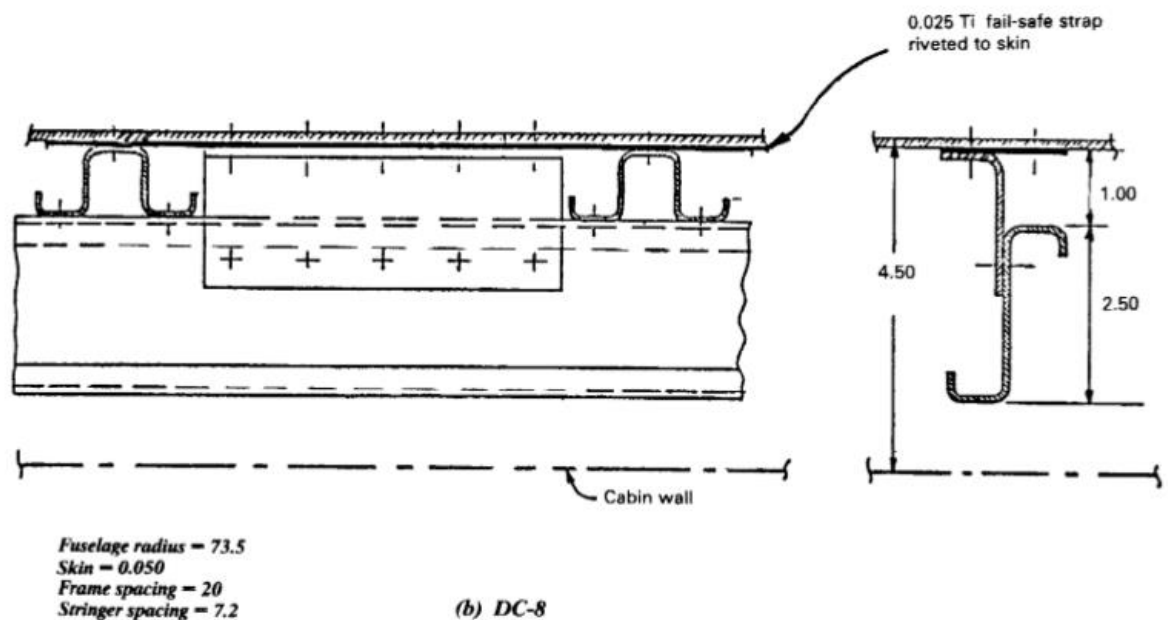


Fig. 1.8 – DC-8. Transport fuselage structural arrangement. (Dimensions in inch and materials are aluminum except as noted)

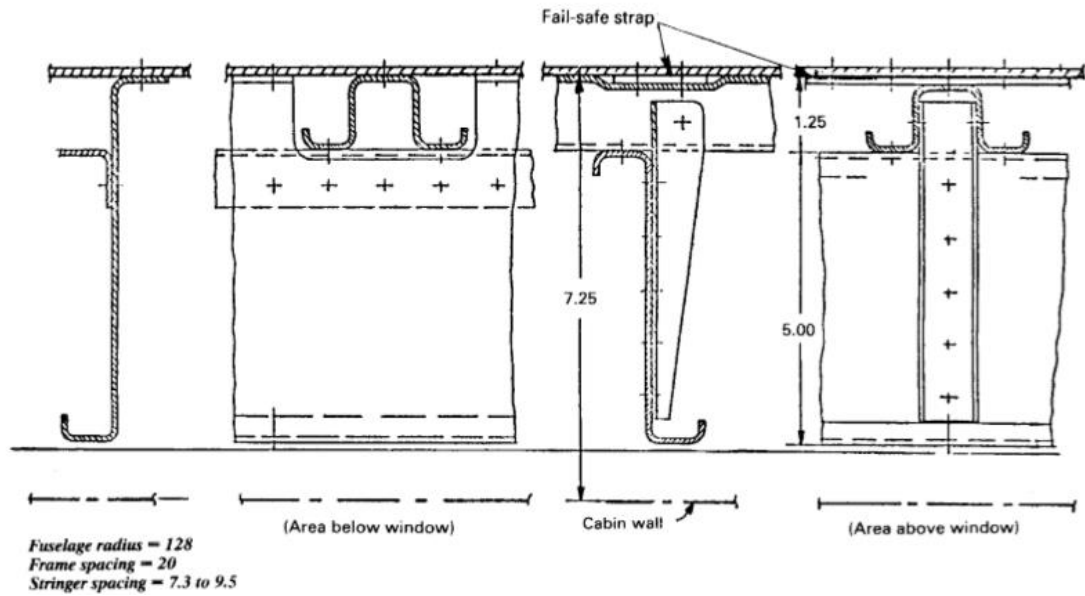


Fig. 1.9 – B747. Transport fuselage structural arrangement. (Dimensions in inch and materials are aluminum except as noted)

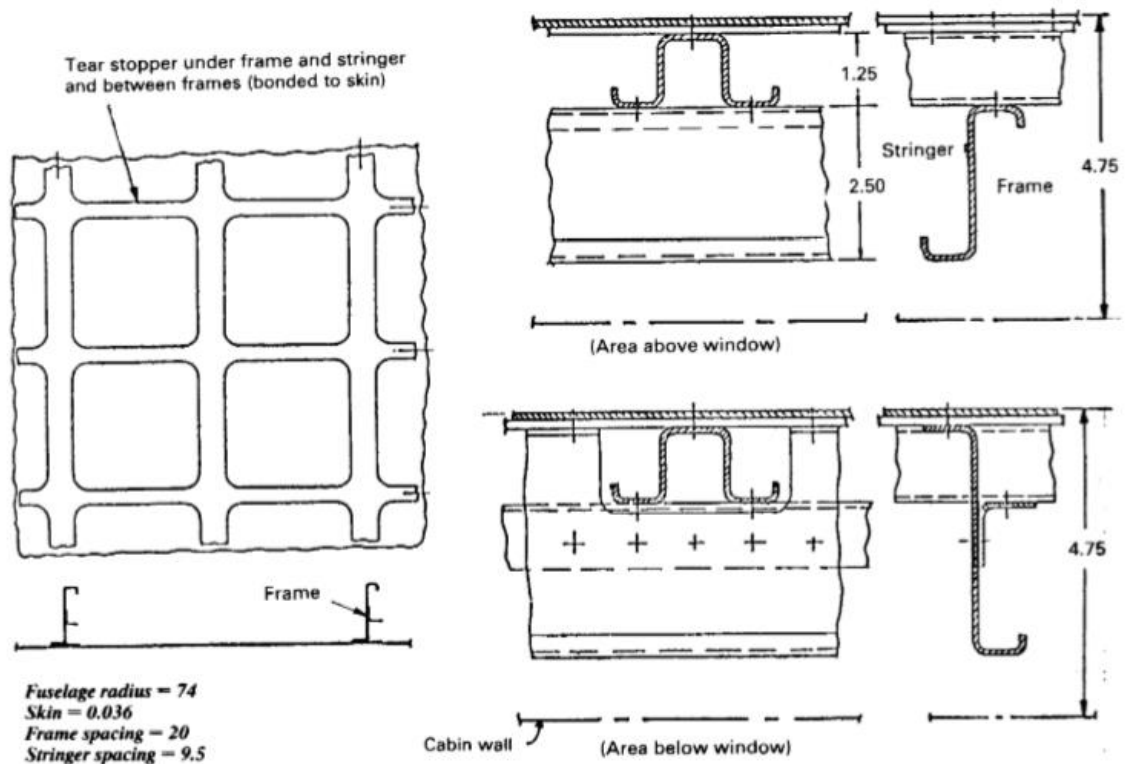


Fig. 1.10 – B737. Transport fuselage structural arrangement. (Dimensions in inch and materials are aluminum except as noted)

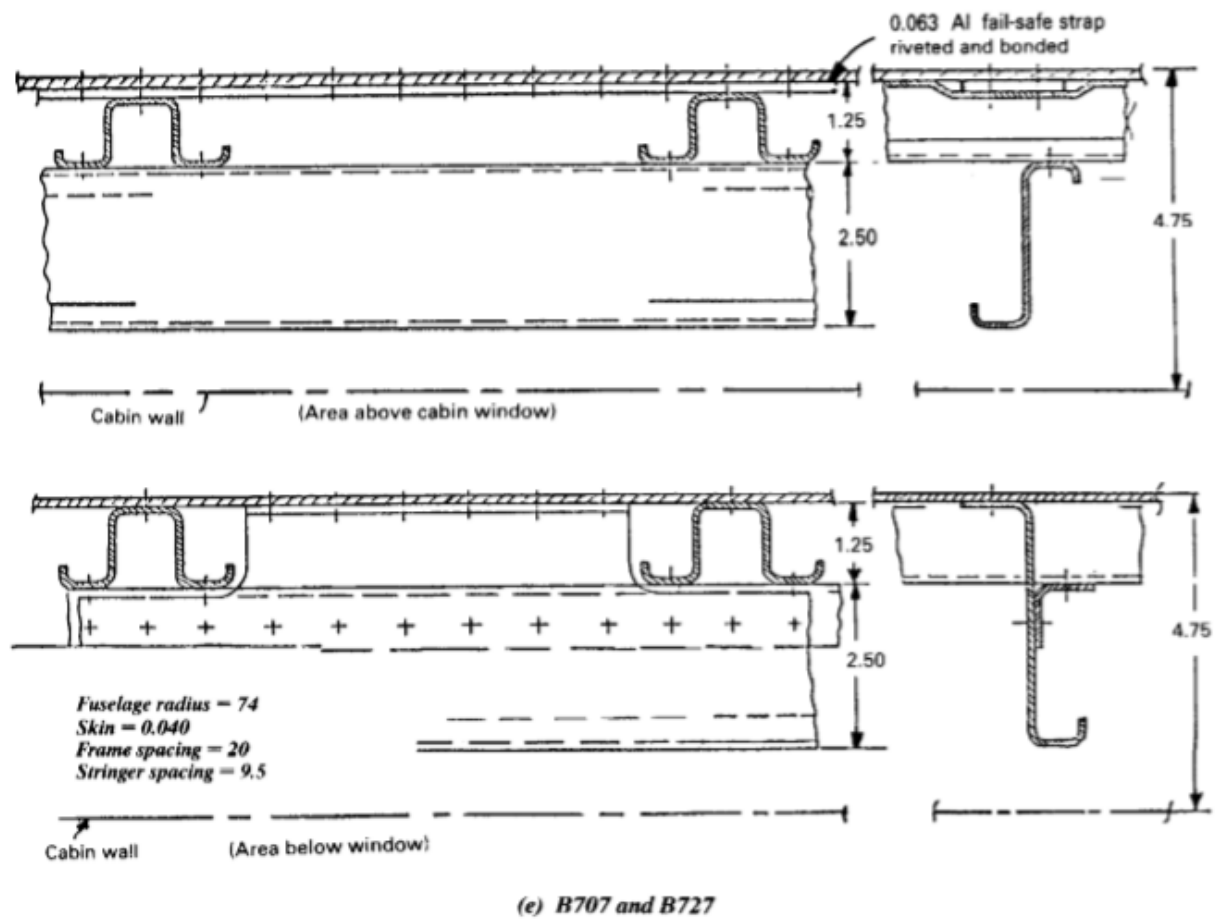


Fig. 1.11 – B707 and B727. Transport fuselage structural arrangement.
(Dimensions in inch and materials are aluminum except as noted)

2. FINITE ELEMENT METHOD. SUBSTANTIATION OF FINITE ELEMENTS TYPE AND SIZE

FEM is a method for solving problems of deformation, thermal conductivity, oscillations, hydrodynamics, electromagnetism, and other problems. Approximation is the replacement of the investigated object with its simplified model. The accuracy of the approximation is the degree of correspondence of the studied object and its simplified model. Discretization is the separation of the model of the studied object into separate (simpler) parts. Finite elements (FE) are separate parts or subdomains into which the model of the object under study is divided. A mesh (mesh model) is a set of interconnected finite elements.

Finite Element Attributes (FE). Attributes are properties of finite elements. The main attributes of FE are: a) The intrinsic dimension (0D, 1D, 2D, 3D) is the number of local spatial coordinates needed to describe the geometry of an element. In dynamic calculation, time is an additional dimension of an element. b) Nodes (nodal points) are characteristic points that describe the geometry of an element and determine its degree of freedom. Nodes can be located at corner (extreme) points, on edges, and inside an element. Elements that have only corner nodes are called linear. Internal nodes can improve the accuracy of the solution, but make the calculation more expensive in terms of computer resources and time. c) Geometry is the geometric form of elements created by its nodes (for example: a segment, a triangle, a prism); d) The degree of freedom is the number of independent parameters characterizing the state of the model under study. e) Defining relations are physical laws that interconnect various parameters (for example, Hooke's law, which relates stress and strain). f) Section properties are the section parameters of an element (for example: area, thickness, moments of inertia) that need to be specified for elements of certain types (for example, for beam elements or flat shell elements).

The main types of FE

- a) Point mass. Attributes: dimension - 0D; 1 node; geometry is a point.
- b) The spring element. Attributes: dimension - 1D; 2 extreme nodes; geometry is

a segment. It is used to approximate springs when solving deformation problems under the action of mechanical stress.

c) The rod element. Attributes: own dimension - 1D; 2 extreme nodes; geometry - a segment; in section properties the area is set. It is used to approximate rods in truss-type structures when solving deformation problems (rods work only in tension and compression), thermal conductivity and vibrations.

d) A beam element. Attributes: dimension - 1D; 2 extreme nodes; geometry - a segment; section properties define the shape, dimensions and orientation. They are used to discretize beam-type structures in solving problems of deformation (tension and compression, bending, torsion, shear or shear), thermal conductivity and vibrations.

e) Elements for discretization of plane bodies and shells, see Fig. 2.1. Attributes: dimension - 2D. They are used in solving problems of deformation, thermal conductivity and oscillations of plane bodies and shells, as well as for modeling two-dimensional or axisymmetric flow of a liquid or gas.

- 1) Triangle. Attributes: 3 corner nodes.

- 2) The quadrangle. Attributes: 4 corner nodes.

- 3) An arbitrary polygon. Attributes: more than 4 corner nodes.

f) Elements for discretization of rigid bodies. Attributes: own dimension - 3D. They are used in solving problems of deformation, thermal conductivity and oscillations of bulk bodies, as well as for modeling three-dimensional flow of liquid and gas.

- 1) Tetrahedron. Attributes: 4 corner nodes.

- 2) The pyramid. Attributes: 5 corner nodes.

- 3) Prism or wedge. Attributes: 6 corner nodes.

- 4) Hexahedron. Attributes: 8 corner nodes.

- 5) An arbitrary polyhedron. Attributes: more than 8 corner nodes.

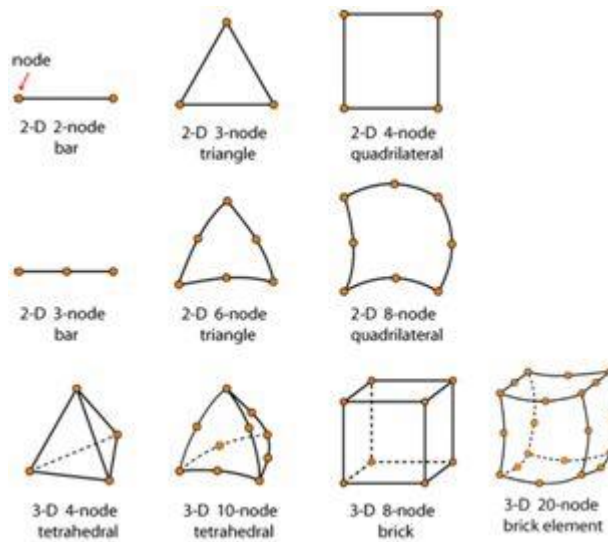


Fig. 2.1 – Types of finite element

During choosing the type and size of the finite element mesh, it is recommended to be guided by the following principles, which will help to obtain results that are as close to reality as possible: regular grid gives more accurate results than irregular; rectangular 4-node FEs give more accurate results than triangular ones; triangular elements with intermediate nodes have accuracy close to a grid of rectangular 4 nodal elements; a rectangular grid with intermediate nodes gives more accurate results than a triangular grid with intermediate nodes, despite the large area; spaces between FEs are not allowed; a combination of triangular and quadrangular elements in one model is allowed; 4-noded elements with a blunt ($> 180^\circ$) internal angle are prohibited.

The number of elements in the calculation model is inversely proportional to the size of the finite elements, while the calculation time increases exponentially with a decrease in the size of the FE.

During choosing the dimensions of a finite element grid, one must always remember that the finite element method itself is a method of numerical approximation, and it cannot give absolute accuracy. Yes, the calculation error decreases with a decrease in the size of the FE, but it cannot be completely eliminated, and often, a further decrease in the network will not lead to a noticeable

increase in accuracy, but the calculation time will increase, this moment must be kept in mind and not aim for too small element size.

Selecting the Finite Element Type.

Use shell elements to model thin structures (where one dimension is much smaller than the other two dimensions). Shell elements decouple the deformation on the surface and the deformation in the normal direction, allowing for a simple and efficient simulation of a thin structure [2].

No firm rules exist to indicate when it is best to use shell elements. If the structure acts as a shell, we can use shell elements [2].

Shell elements are usually created by meshing a surface representing the position and shape of the structure, and associating them with section data to represent shell thickness and properties [2].

Axisymmetric shell elements are available for efficiently modeling axisymmetric shell structures in 2-D. Axisymmetric shell elements, such as SHELL208 and SHELL209, may include out-of-plane translational degrees of freedom to model uniform torsion about the axis [2].

Use beam or link (truss) elements to represent relatively long, thin pieces of structural continua (where two dimensions are much smaller than the other dimension) [2].

Beam elements assume the direct stresses in the nonaxial direction to be zero, and ignore the deformations in the nonaxial directions (although cross sections can be scaled in a nonlinear analysis). For link elements, shear stress, stress gradients, and deformation are also ignored [2].

Simulating beam/truss structures is relatively simple, as elements can be created by meshing lines or curves representing the position and shape of the structure (rather than by meshing the volumes). The elements are associated with section data to represent the beam or truss cross sections [2].

Although beams can be used in cases where link/truss elements suffice, they are not as computationally efficient as link elements [2].

The available beam element types are: BEAM188, a 3-D two-node beam and BEAM189, a 3-D three-node beam [2].

Both elements are based on Timoshenko beam theory which includes shear-deformation effects. The elements can account for unrestrained warping and restrained warping of cross-sections [2].

Because bending resistances, element loads, or both are normally different in the y and z directions, a beam element must be oriented about its own axis. Typically, the orientation is accomplished via an orientation node, a node not lying on its own axis. If bending resistance is negligible, consider using LINK180, a 3-D spar (or truss) [2].

The skins, webs and caps of stringer and frame are set by 2D elements SHELL209 due to small thickness and loading. For modelling of fastener is used 1D element – BEAM188 element.

BEAM188 Element Description

BEAM188 is suitable for analyzing slender to moderately stubby/thick beam structures. The element is based on Timoshenko beam theory which includes shear-deformation effects. The element provides options for unrestrained warping and restrained warping of cross-sections [2].

The element is a linear, quadratic, or cubic two-node beam element in 3-D. BEAM188 has six or seven degrees of freedom at each node. These include translations in the x, y, and z directions and rotations about the x, y, and z directions. A seventh degree of freedom (warping magnitude) is optional. This element is well-suited for linear, large rotation, and/or large strain nonlinear applications [2].

The element includes stress stiffness terms, by default, in any analysis with large deflection. The provided stress-stiffness terms enable the elements to analyze flexural, lateral, and torsional stability problems (using eigenvalue buckling, or collapse studies with arc length methods or nonlinear stabilization) [2].

Elasticity, plasticity, creep and other nonlinear material models are supported. A cross-section associated with this element type can be a built-up section referencing more than one material. Added mass, hydrodynamic added mass and loading, and buoyant loading are available [2].

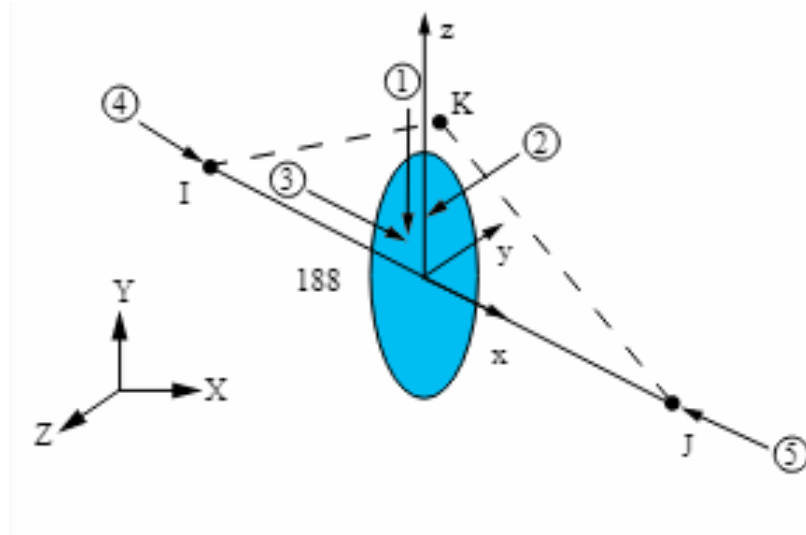


Fig. 2.2 – BEAM188 Geometry

For more detailed information about this element, see BEAM188 - 3-D 2-Node Beam in the Theory Reference [2].

SHELL209 Element Description

The SHELL209 element is suitable for analyzing thin to moderately thick axisymmetric shell structures. It is a three-node element with three degrees of freedom at each node: translations in the X, Y directions, and a rotation about the Z-axis. A fourth translational degree of freedom in z direction can be included to model uniform torsion (KEYOPT(2) = 1). When the membrane option is used, the rotational degree of freedom is excluded. (For higher efficiency, the two-node element SHELL208 may be more suitable.) [2].

The element is well suited for linear, large rotation, and/or large strain nonlinear applications. Changes in shell thickness and follower effects of distributed pressures

are accounted for in nonlinear analyses, and it can be used for layered applications for modeling laminated composite shells or sandwich construction. See SHELL209 for more details about this element [2].

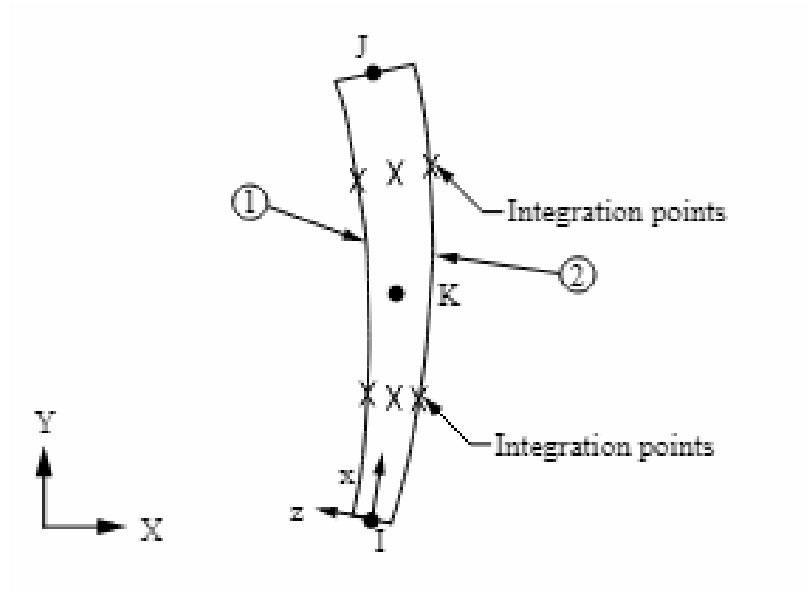


Fig. 2.3 – SHELL209 Geometry

3. ANALYSIS OF THE LONGITUDIAL LAP SPLICE OF SKINS USING FINITE ELEMENT METHOD

Formulation of the problem

Study the influence of rigidity and layout of the fuselage frames on the distribution of forces in the skin longitudinal lap splice is performed separately. In the first case it is evaluated influence of the frame layout, in the second case – the influence of frame rigidity.

Loading

For both analysis cases as a load is internal pressure loading.

The fuselage internal pressure depends on the cruise altitude and the comfort desired for the occupants. Pressure differential may be readily determined from the altitude charts, if the actual altitude and the desired cabin pressure are known [1].

Fuselage pressurization is an important structural loading. It induces hoop and longitudinal stresses in the fuselage which must be combined with flight and ground loading conditions. The important consideration for establishing the fuselage design pressures is the cabin pressure differential of the present in altitude and the fuselage is designed to maintain: provide an 8,000 ft. altitude cabin pressure; the max flight altitude at 43,000 ft.; than the pressures used with flight and ground conditions on the airplane are [1]:

Table 3.1 – Internal pressure

Condition	Max. Positive Pressure	Max. Negative Pressure
Flight (All)	8.85 psi	-0.50 psi
Ground	1.0 psi	-0.50 psi

In addition, a pressure of $1.33 \times 8.85 = 11.77$ psi is considered to act alone.

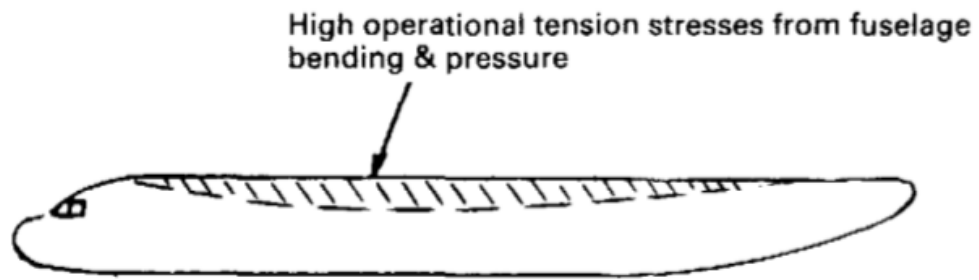


Fig. 3.1 – The most loaded part of fuselage

Properties of longitudinal lap splice

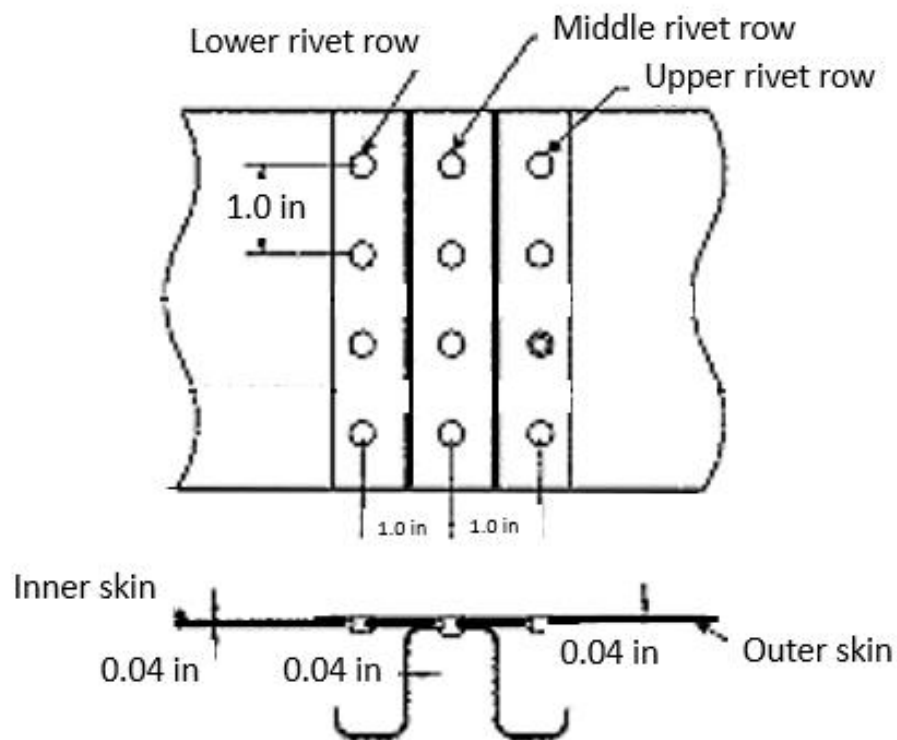


Fig. 3.2 – Typical riveted lap splice used for study

A typical riveted lap splice of fuselage skins is used for study, see Fig. 3.2 above. Fasteners are 6/32-inch diameter rivets from 2117-T3, 1-inch pitch and spacing, 0.5 inch of edge distance.

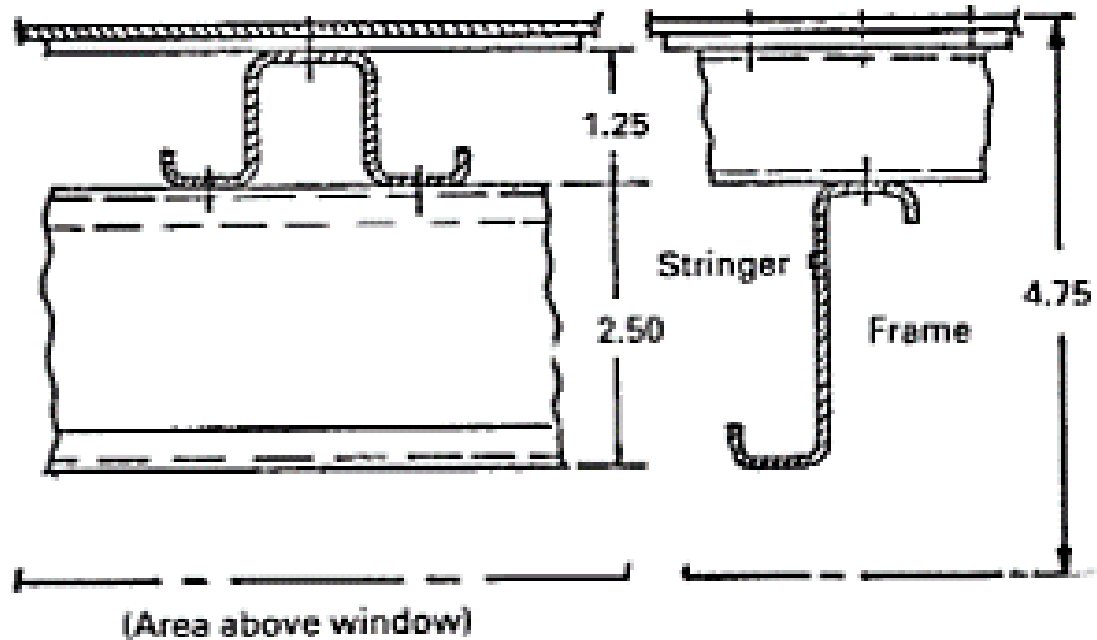


Fig. 3.3 – Fuselage geometrical properties

During analysis, in the first case it is changed a frame spacing as a factor of frame layout, and in the second case – a frame thickness as a factor of frame rigidity. All other properties are fixed.

Table 3.2 – Properties of analysed structure

Radius Rmax	Skin thickness	Str. thickness	Typical Frame thickness	Material Skin/Str	Str. Spacing	Typical Frame Spacing
74 in	0.04 in	0.04 in	0.05 in	2024-T3/ 7075-T6	9.0 in	20 in

The influence of frame layout on the distribution of forces in the skin longitudinal lap splice

For analysis influence of frame layout it is used typical geometric parameters that are shown above.

Table 3.3 – Properties of analysed structure

Case	Skin thickness	Str. thickness	Frame thickness	Material Skin/Str	Str. Spacing	Frame Spacing
1	0.04 in	0.04 in	0.05 in	2024-T3/ 7075-T6	9.0 in	20 in
2	0.04 in	0.04 in	0.05 in	2024-T3/ 7075-T6	9.0 in	22 in
3	0.04 in	0.04 in	0.05 in	2024-T3/ 7075-T6	9.0 in	24 in
4	0.04 in	0.04 in	0.05 in	2024-T3/ 7075-T6	9.0 in	26 in
5	0.04 in	0.04 in	0.05 in	2024-T3/ 7075-T6	9.0 in	28 in
6	0.04 in	0.04 in	0.05 in	2024-T3/ 7075-T6	9.0 in	30 in

For study of influence of frame layout on the distribution of forces in the skin longitudinal lap splice the spacing of frames are changing from 20 inches to 30 inches with increment of 2.0 inches. Another words – the spacing is changing with increment of 10% of primary 20-inch spacing that is tpical for most airplains.

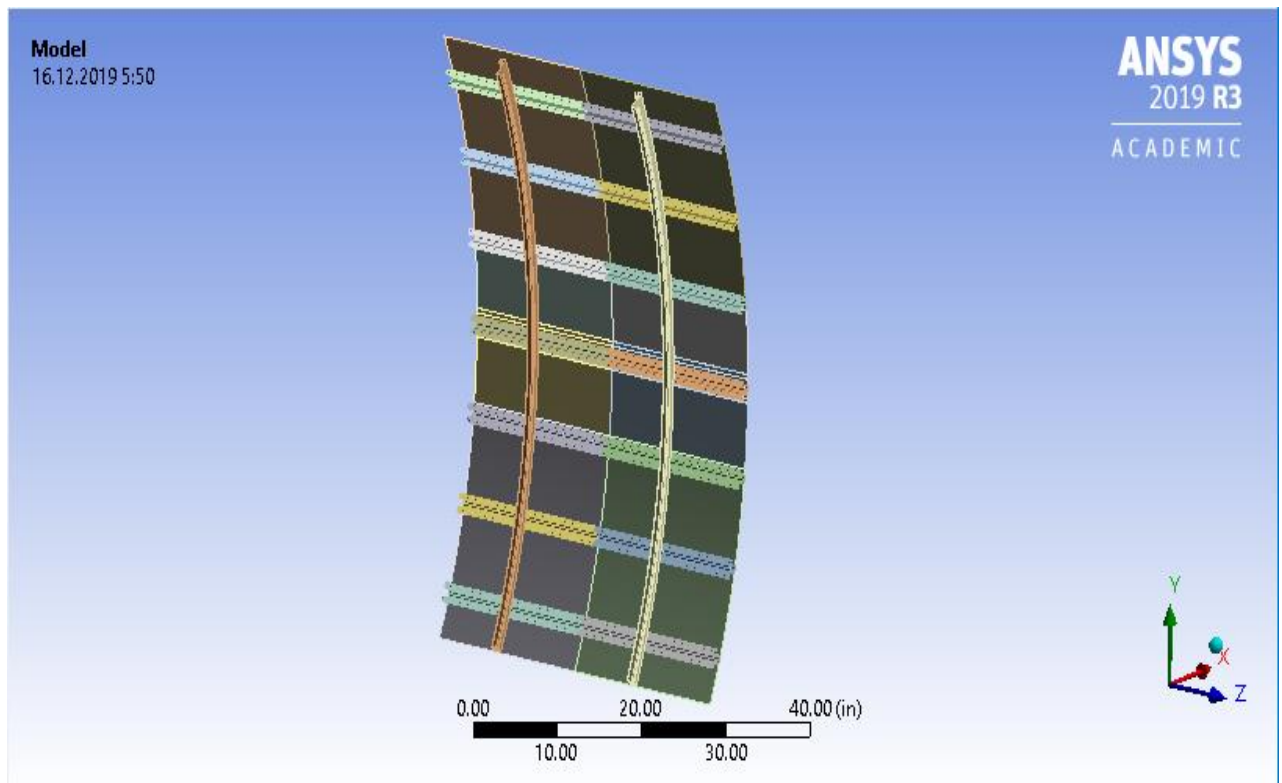


Fig. 3.4 – Geometrical model of 20-inch frame spacing

The analysis model is created and prepared using SpaceClaim (ANSYS2019R3 Academic).

All structural parts are modelled as shells. It is acceptable due to small thicknesses and loading. It is modelled one bay between two frames and a half of bay from both sides of frames.

To creations geometric models for other analysis cases, it is disided to make assembly of two parts of a half-bay-and-frame structure and a nessecary numbers of

2-inch width pieces of fuselage structure between frames to create necessary spacing between them. It is easy to re-build model by this way, see Fig. 3.5 – 3. below.

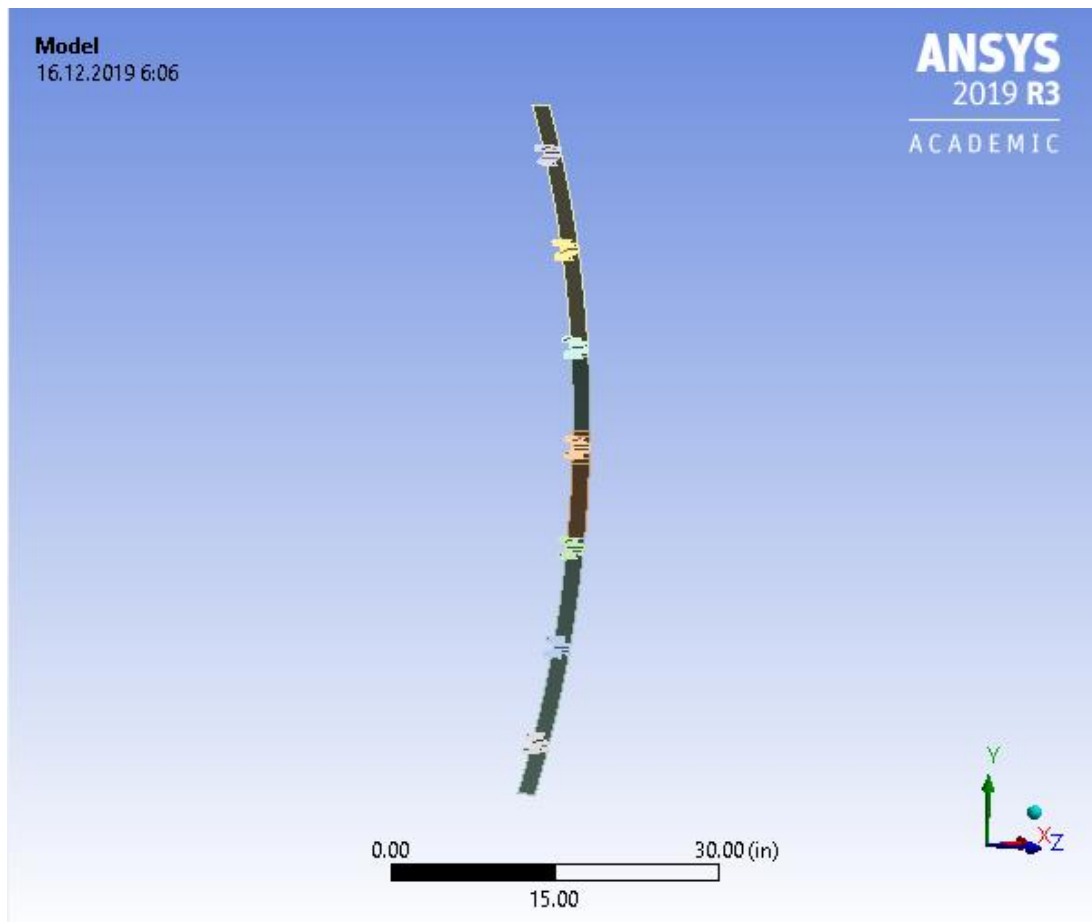


Fig. 3.5 – 2-inch piece of fuselage structure

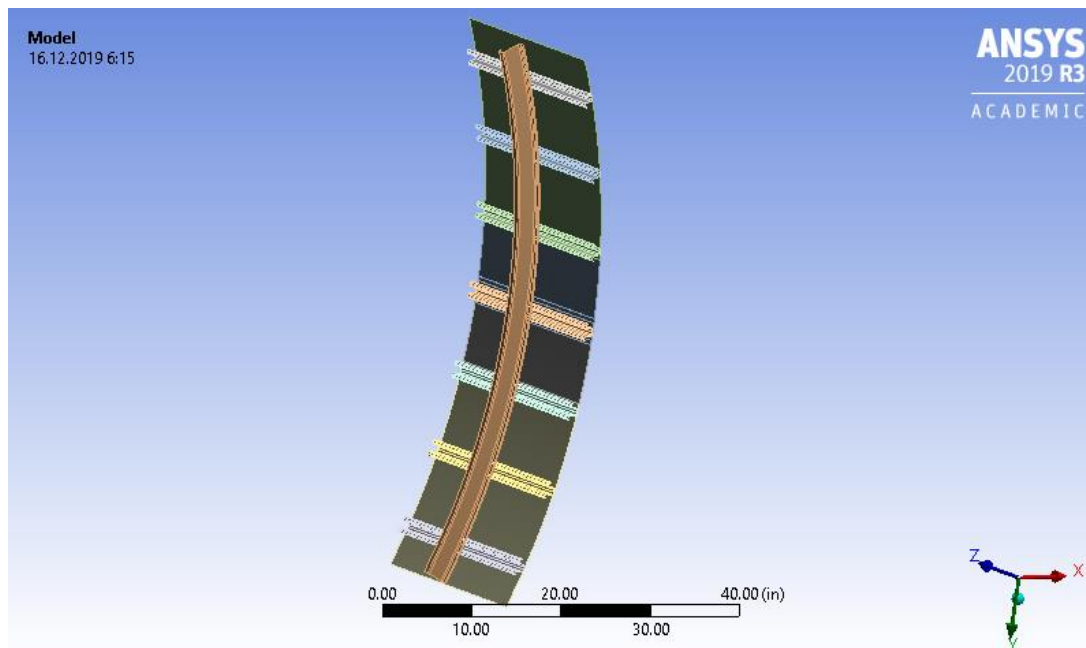


Fig. 3.6 – A half-bay-and-frame structure

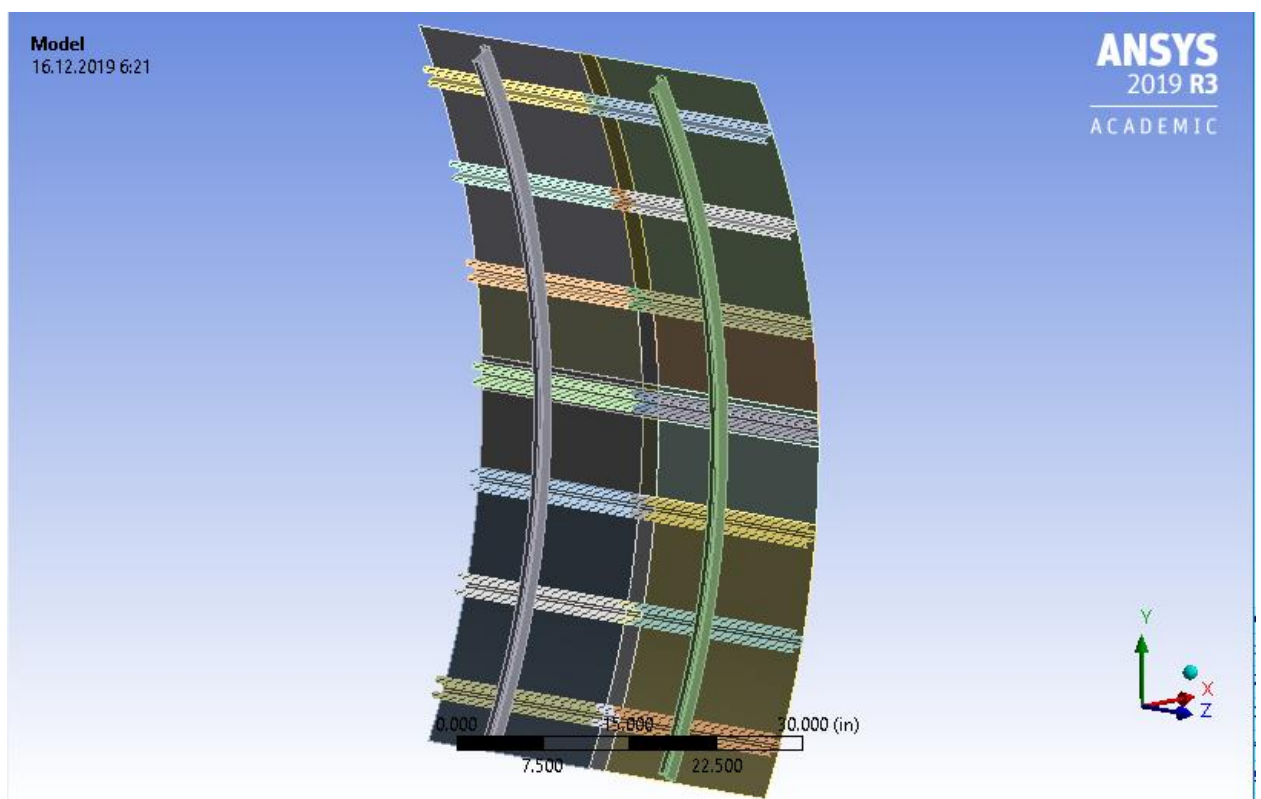


Fig. 3.7 – Geometrical model of 22-inch frame spacing

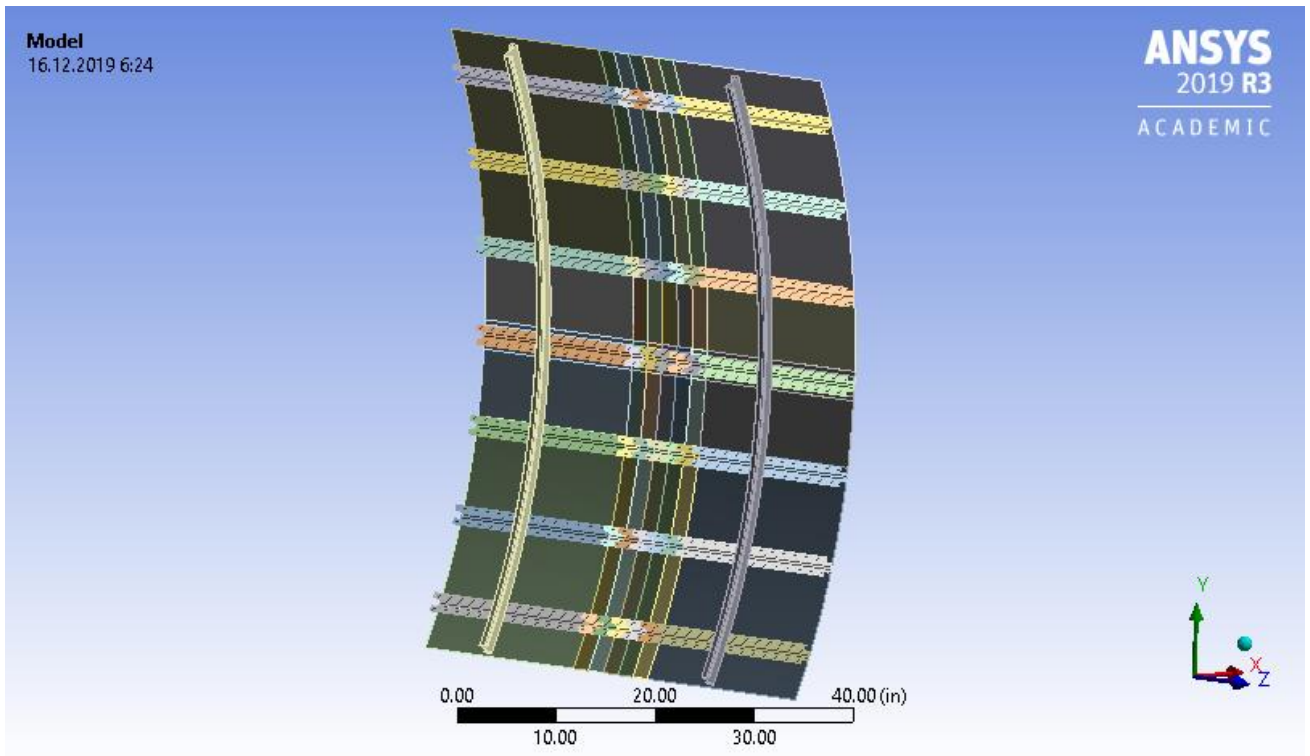


Fig. 3.8 – Geometrical model of 30-inch frame spacing

Mesing

When the all geometric models are created the next step is meshing. It is used standard mesher in Mechanical module. Number of elements on 20-inch frame spacing model is 10,850 and on 30-inch frame spacing model is 14,030 that does not increase limit for ANSYS2019R3 projects.

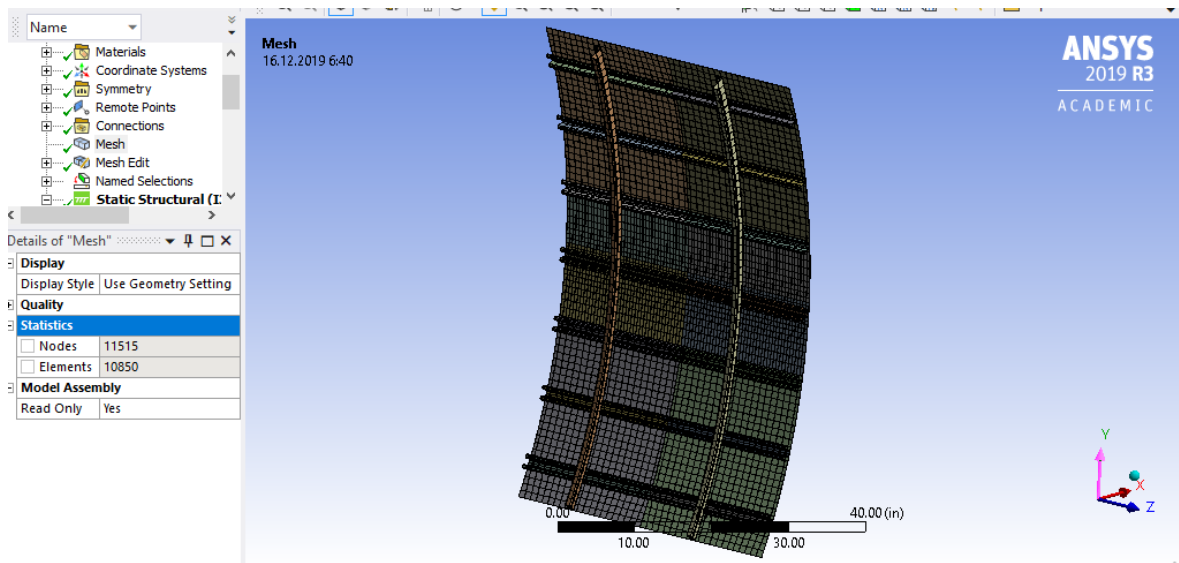


Fig. 3.9 – Mesh on 20-inch frame spacing model

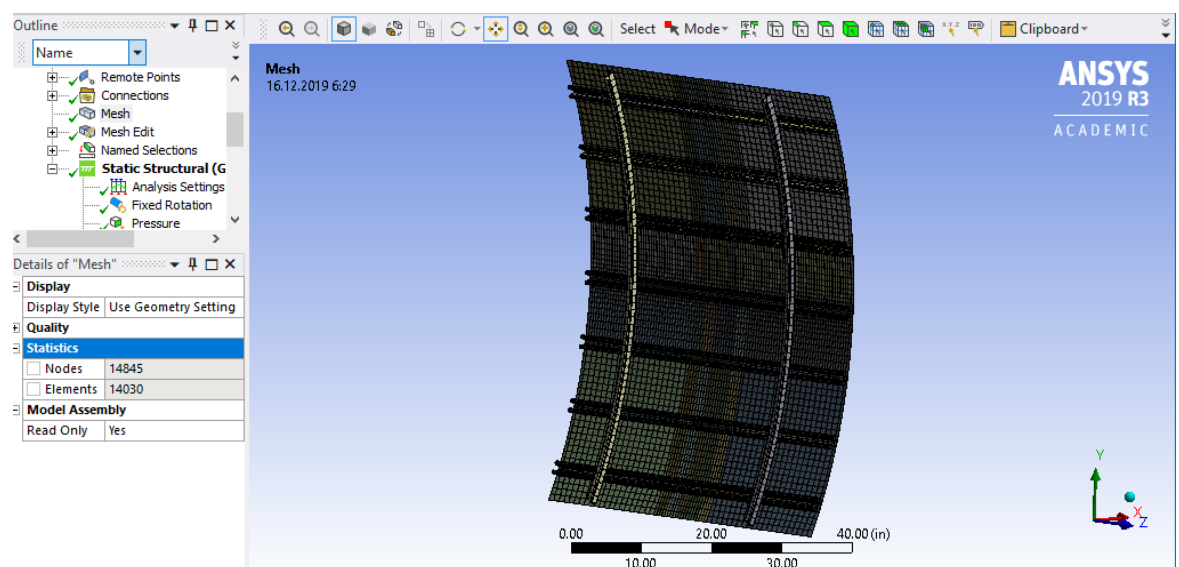


Fig. 3.10 – Mesh on 30-inch frame spacing model

Element size at analysed area – lap splice – is smaller, see figures above.

Also mesh quality is analysed – element size and types. mostly element type is Quad4 , this is a good indicator for the accuracy of the solution.

Details of "Mesh"	
[-] Display	
Display Style	Use Geometry Setting
[-] Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Aggressive Mechanical
Mesh Metric	Element Quality
<input type="checkbox"/> Min	9.5057e-002
<input type="checkbox"/> Max	0.99947
<input type="checkbox"/> Average	0.71459
<input type="checkbox"/> Standard Deviation	0.27236
[-] Statistics	
<input type="checkbox"/> Nodes	13513
<input type="checkbox"/> Elements	12758
[-] Model Assembly	
Read Only	Yes

Fig. 3.11 – Mesh quality – elements size

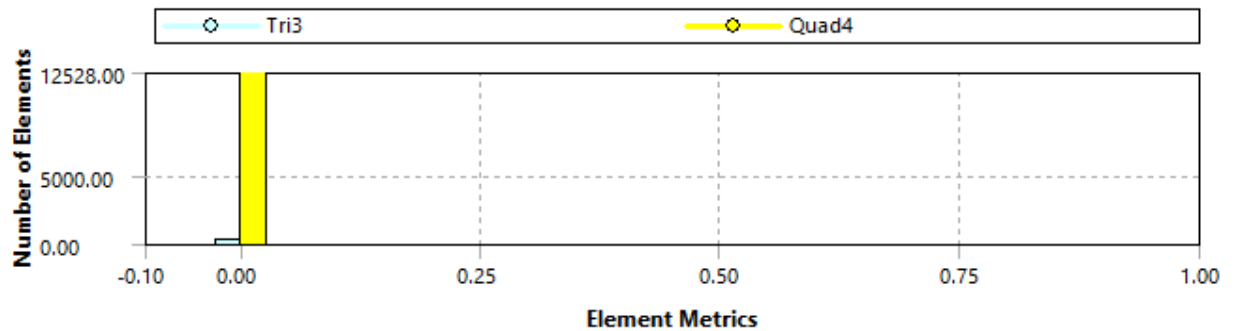


Fig. 3.12 – Mesh quality – elements types

Model properties

Material properties and shell thicknesses are applied, see figures below.

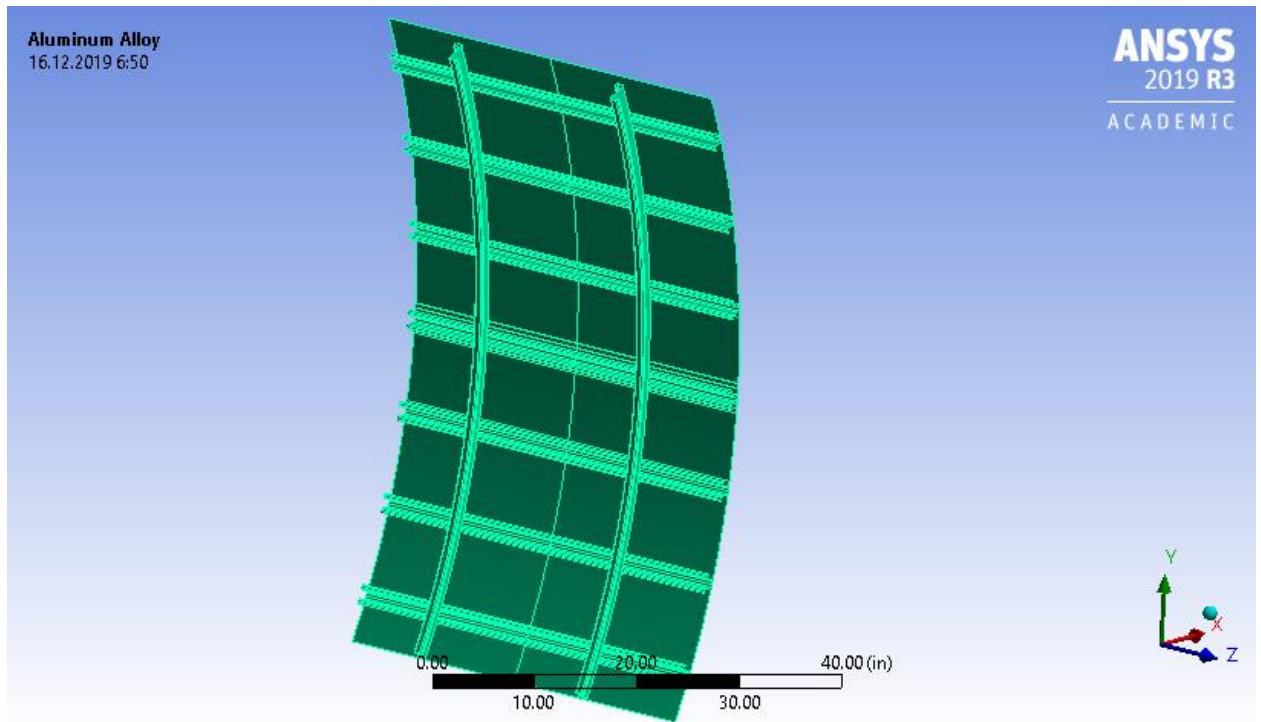


Fig. 3.13 – Material properties

Graphics Properties	
Definition	
Suppressed	No
Dimension	3D
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate Sys...
Reference Temperature	By Environment
Thickness	5.e-002 in
Thickness Mode	Manual
Offset Type	Middle
Treatment	None
Material	
Assignment	Aluminum Alloy
Nonlinear Effects	Yes
Thermal Strain Effects	Yes

Fig. 3.14 – Shell thickness

Boundary conditions and loading

To obtain more accuracy solution and create more accuracy analysis model, a symmetry condition is applied for three free edges of model – three symmetry regions, see Fig. 3.15.

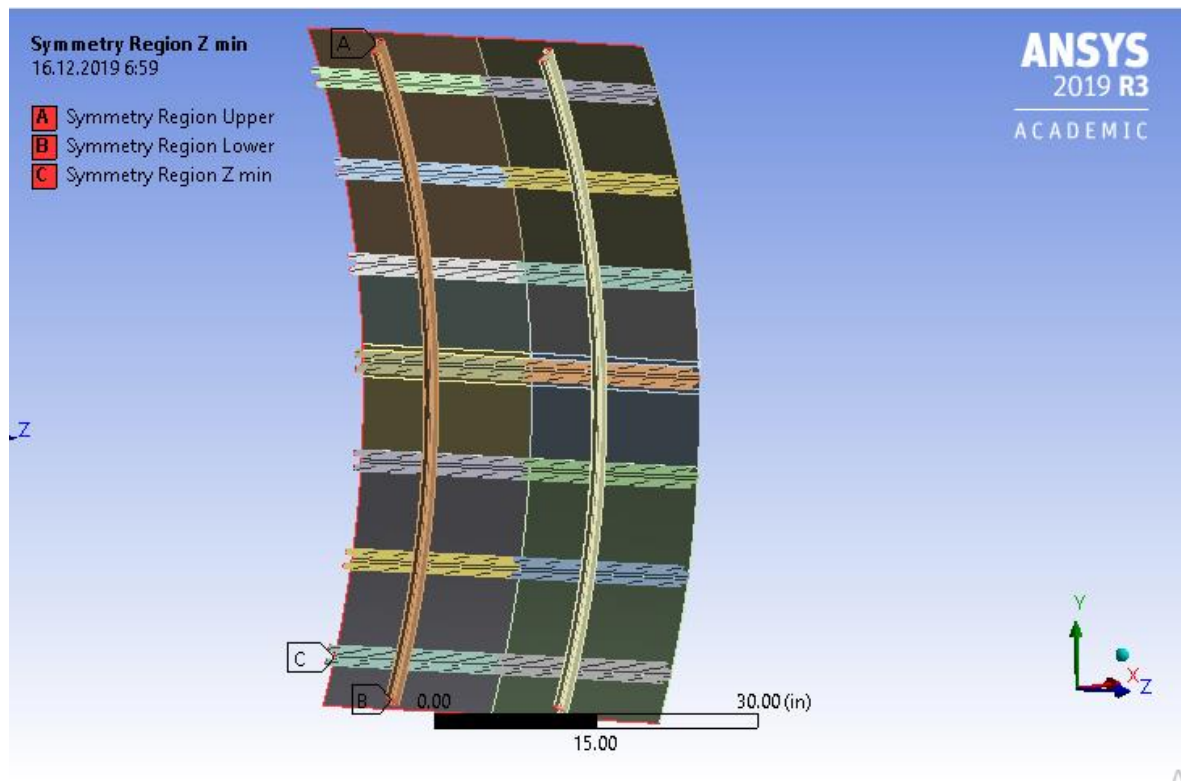


Fig. 3.15 – Symmetry condition

Riveted joints are modelled by beam elements BEAM188 that have properties a 6/32-inch diamener rivet from 2117-T3, see Fig. 3.16.

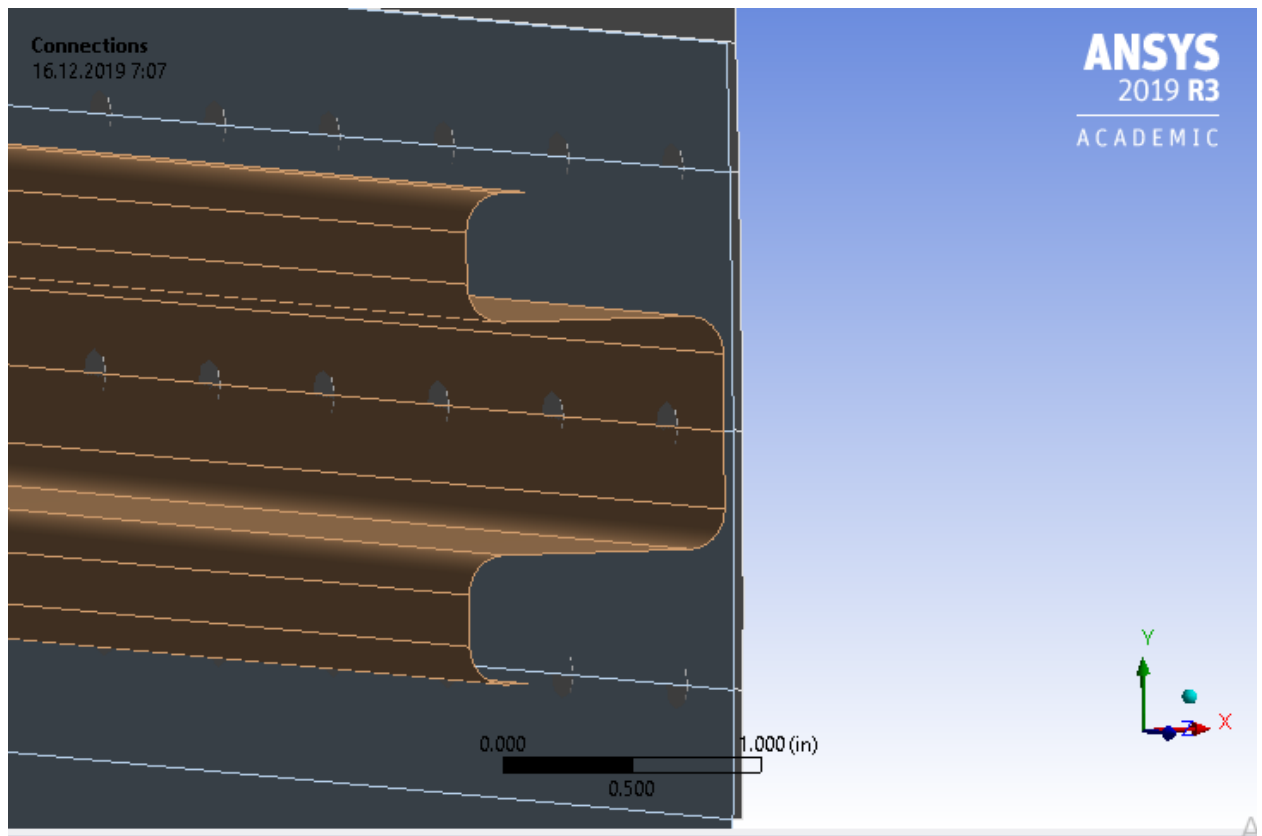


Fig. 3.16 – Riveted joints – BEAM188 elements

One free edge of model is constrained – fixed rotation around X and Y axes in global coordination system. And the same free edge is loaded by longitudinal load that is created by internal pressure, see Fig. 3.17 – 3.18.

The internal pressure is applied to inner sides of skins, see Fig. 3.19.

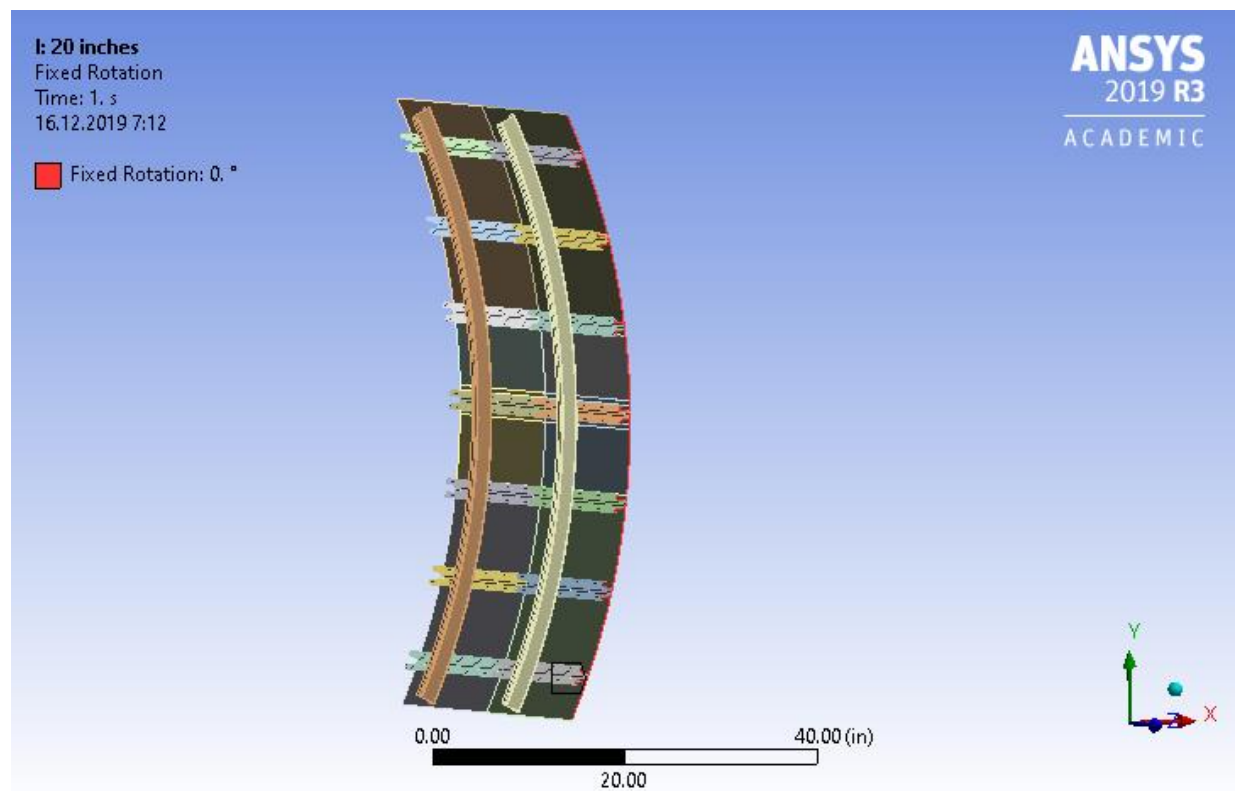


Fig. 3.17 – One free edge is constrained to rotate around X and Y axes

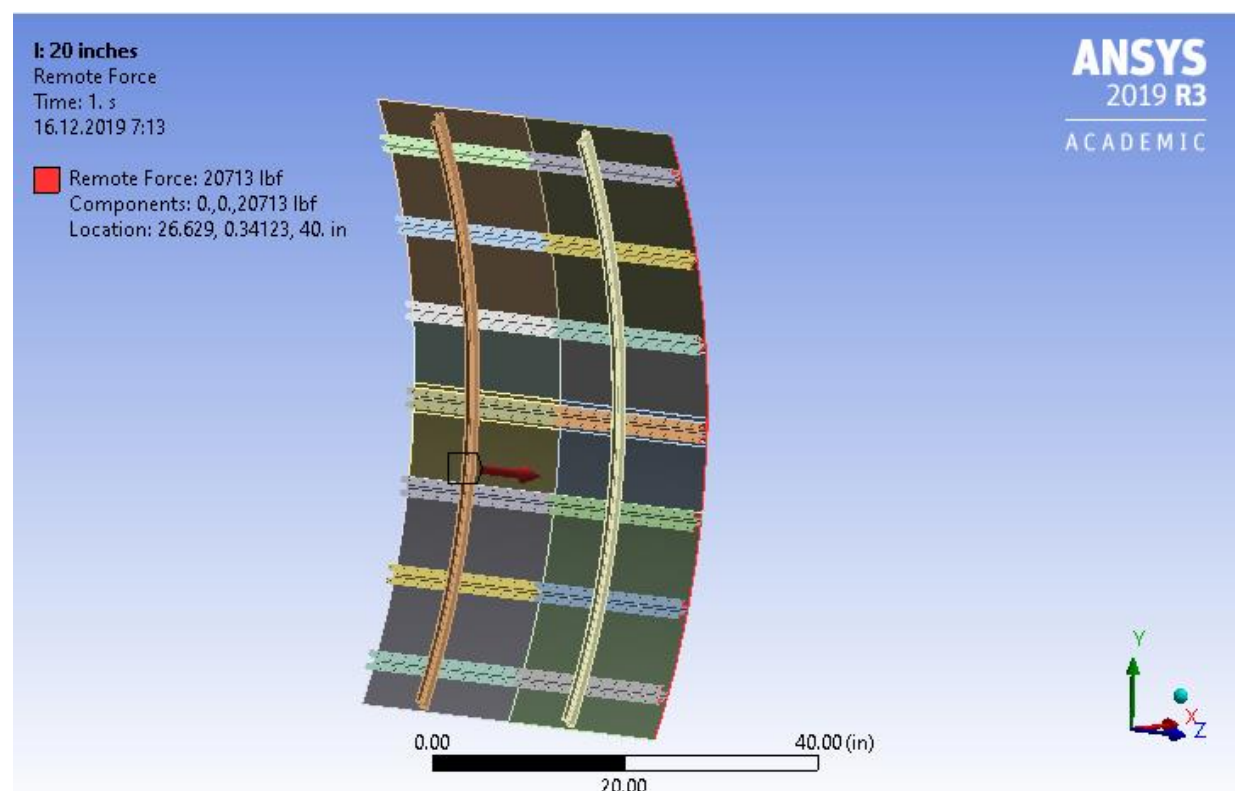


Fig. 3.18 – One free edge is loaded by longitudinal load

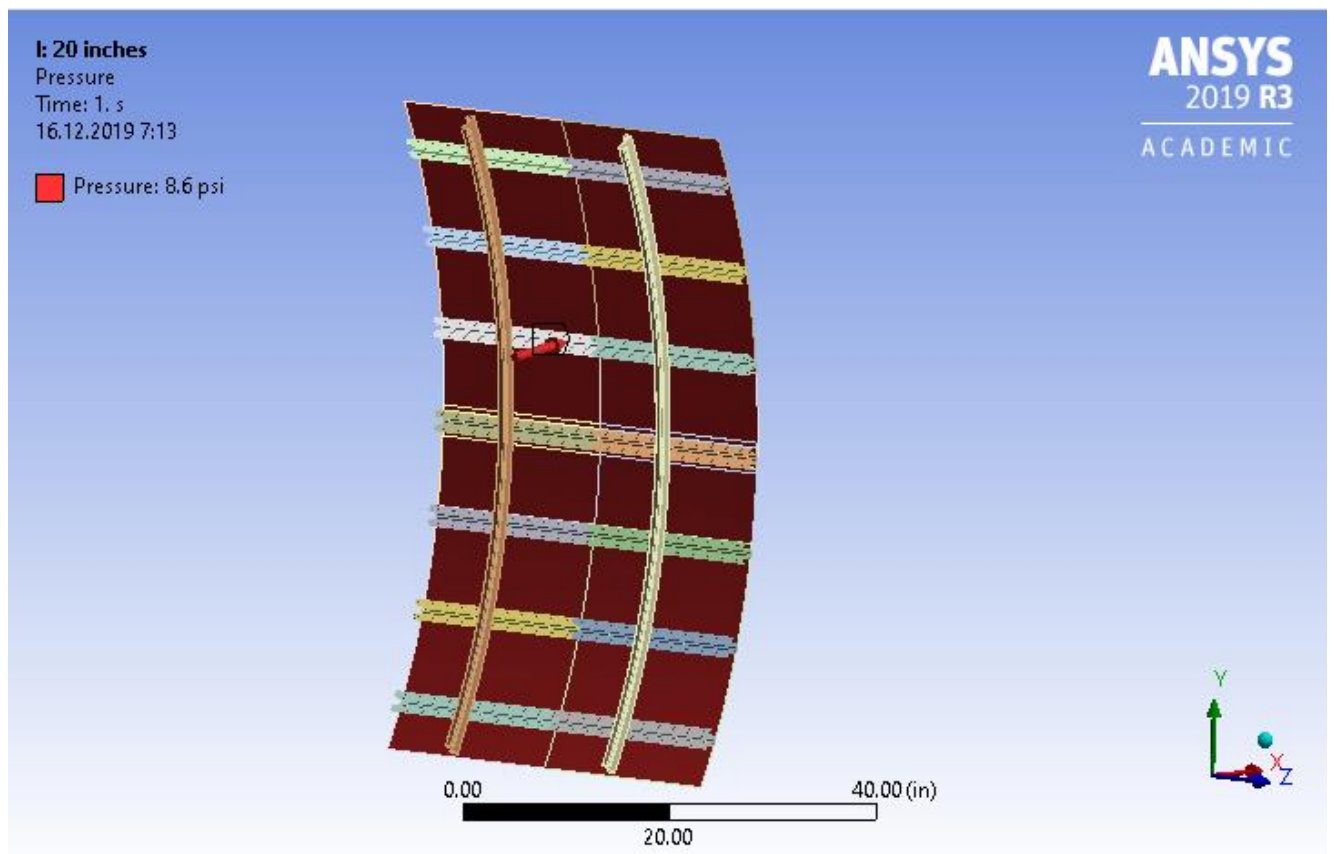


Fig. 3.19 – Inner side of skin under internal pressure

Stress-Strain-States of all analysis cases

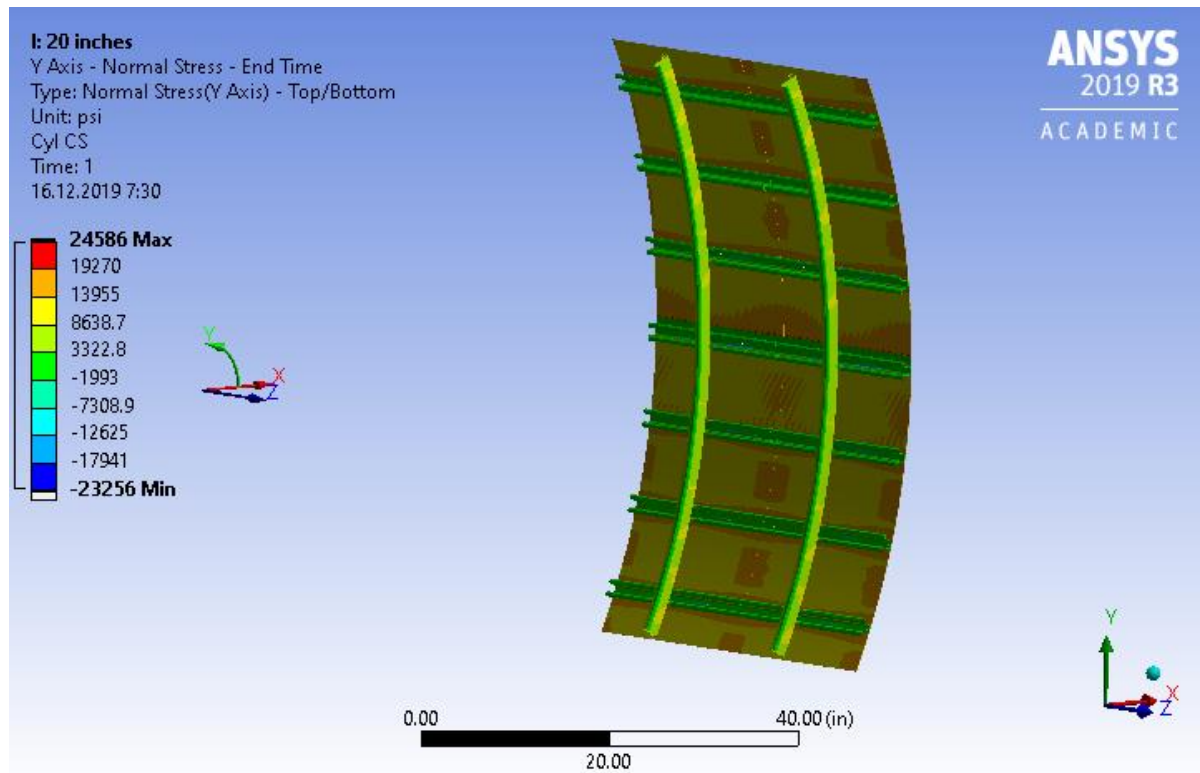


Fig. 3.20 – Y-axis Normal Stresses – 20-inch frame spacing model

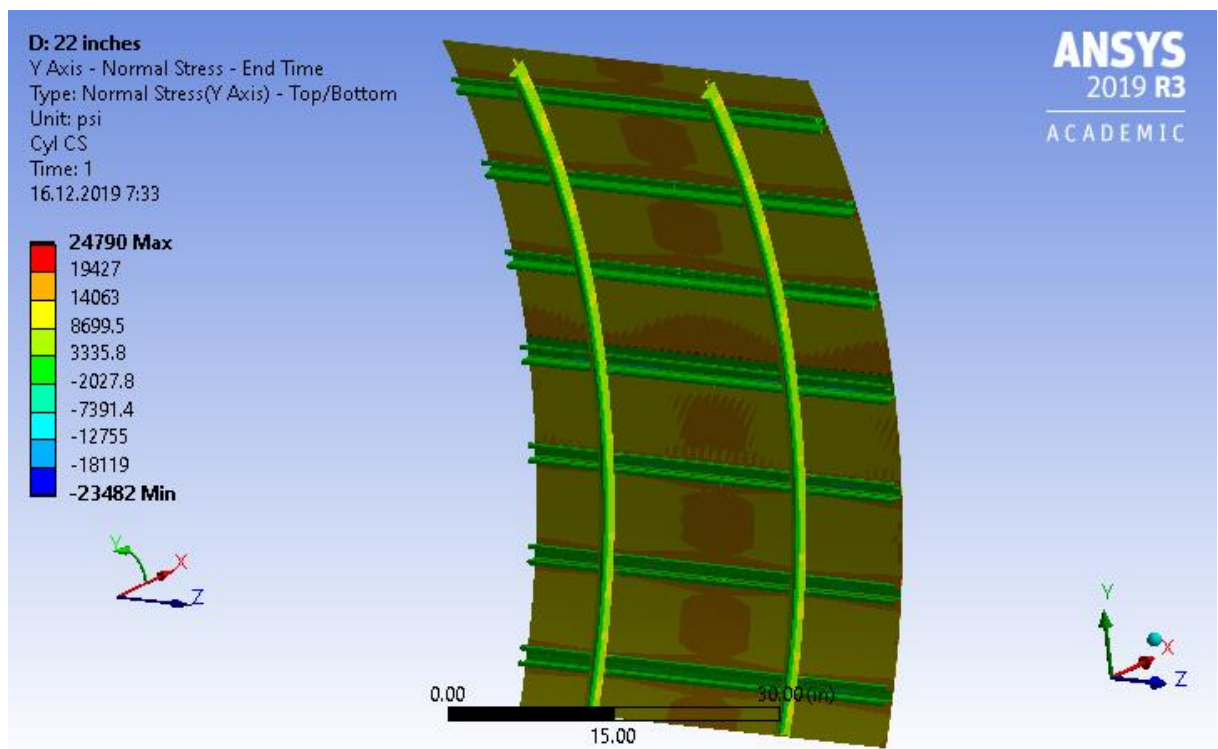


Fig. 3.21 – Y-axis Normal Stresses – 22-inch frame spacing model

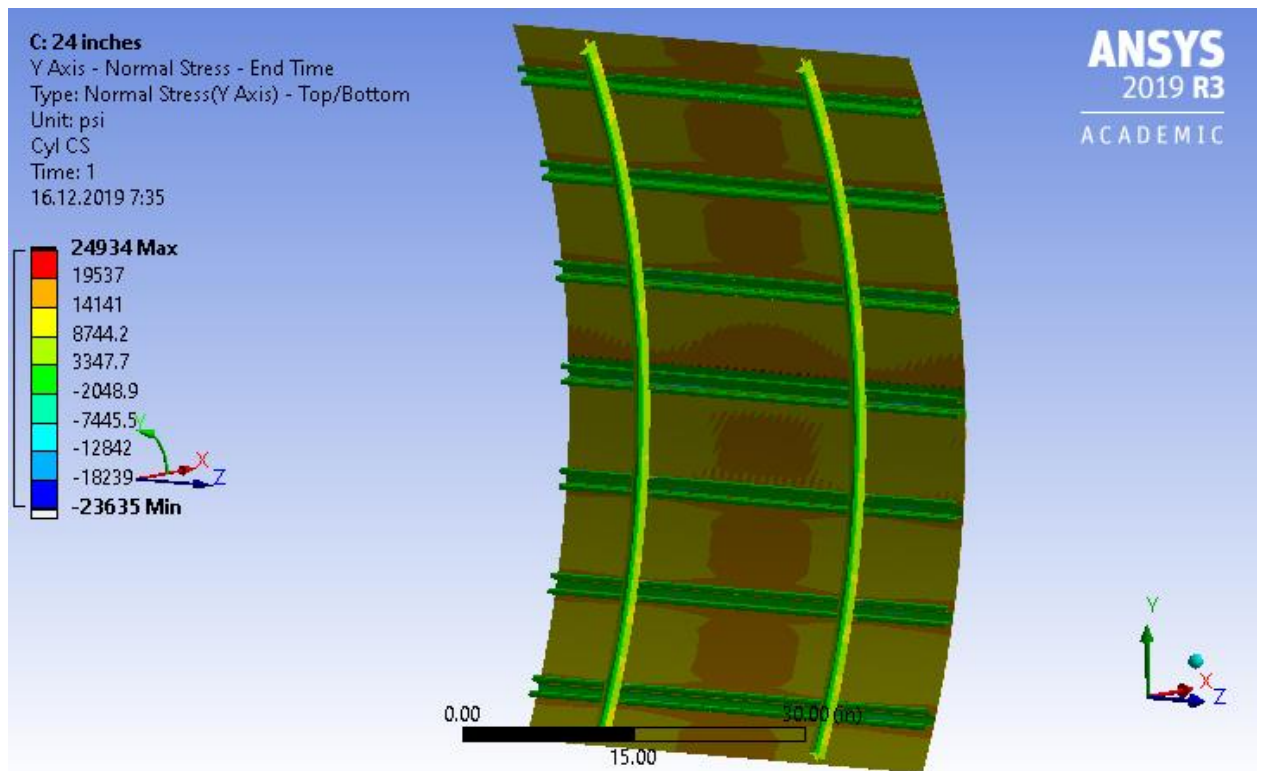


Fig. 3.22 – Y-axis Normal Stresses – 24-inch frame spacing model

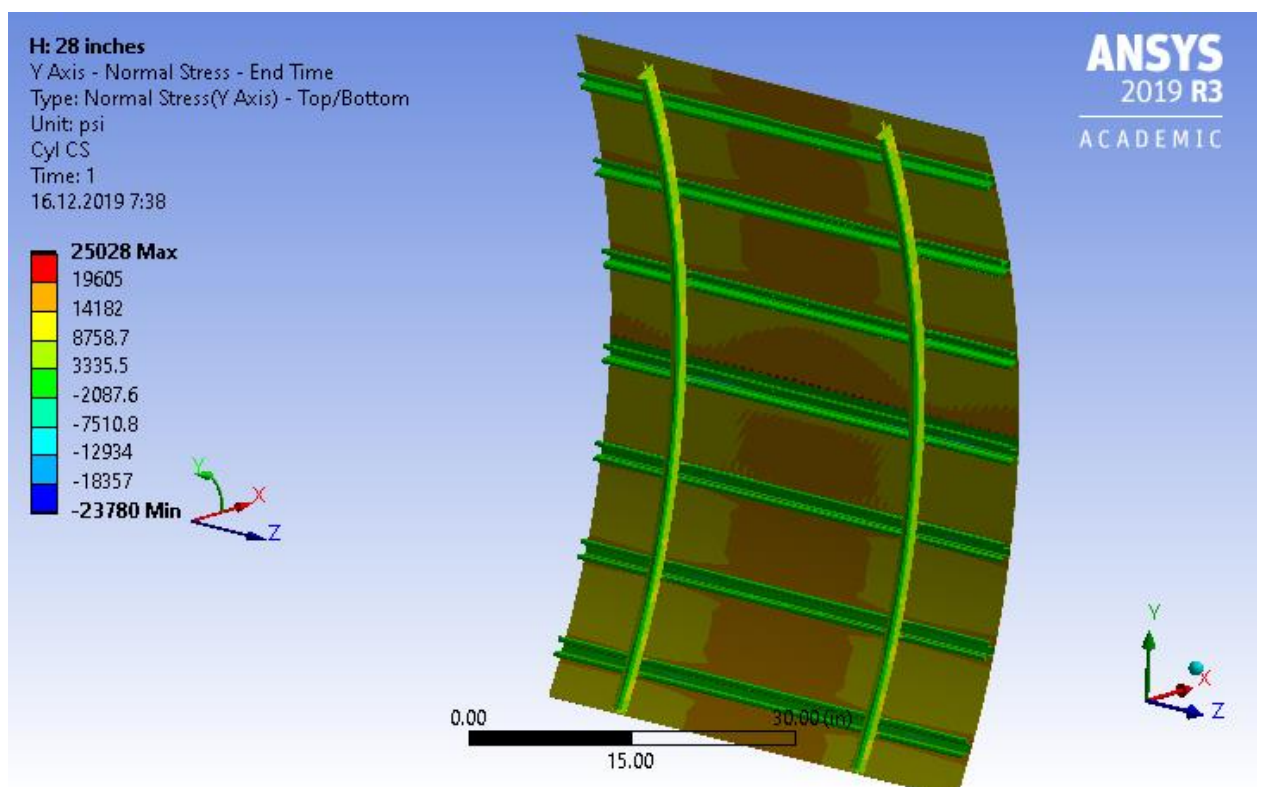


Fig. 3.23 – Y-axis Normal Stresses – 28-inch frame spacing model

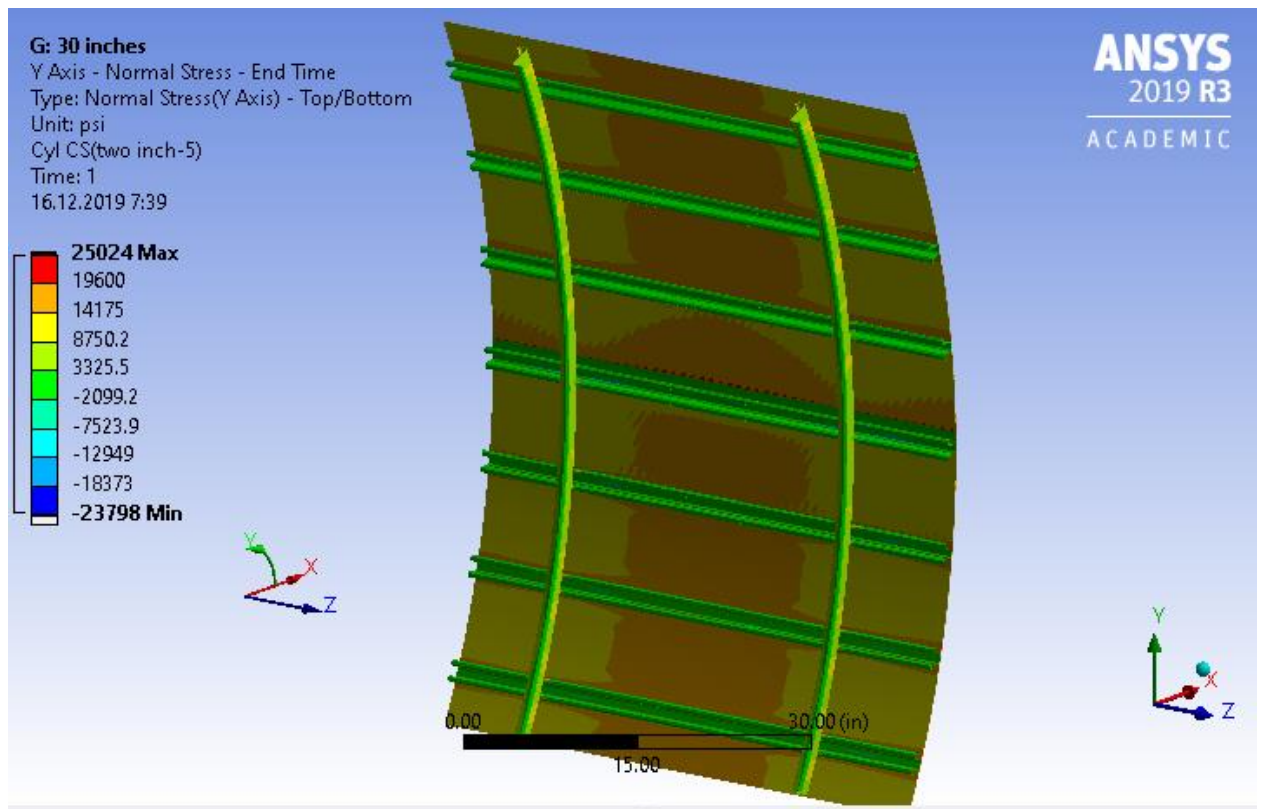


Fig. 3.24 – Y-axis Normal Stresses – 30-inch frame spacing model

The influence of frame rigidity on the distribution of forces in the skin longitudinal lap splice

For analysis influence of frame rigidity it is used typical geometric parameters that are shown below.

Table 3.4 – Properties of analysed structure

Case	Skin thickness	Str. thickness	Frame thickness	Material Skin/Str	Str. Spacing	Frame Spacing
1	0.04 in	0.04 in	0.05 in	2024-T3/ 7075-T6	9.0 in	20 in
2	0.04 in	0.04 in	0.055 in	2024-T3/ 7075-T6	9.0 in	20 in
3	0.04 in	0.04 in	0.06 in	2024-T3/ 7075-T6	9.0 in	20 in
4	0.04 in	0.04 in	0.065 in	2024-T3/ 7075-T6	9.0 in	20 in
5	0.04 in	0.04 in	0.07 in	2024-T3/ 7075-T6	9.0 in	20 in
6	0.04 in	0.04 in	0.075 in	2024-T3/ 7075-T6	9.0 in	20 in

For study of influence of frame rigidly on the distribution of forces in the skin longitudinal lap splice the thickness of frames are changing from 0.05 inches to 0.075 inches with increment of 0.005 inches. Another words – the thickness is changing with increment of 10% of primary 0.05-inch thickness that is tpical for most airplains.

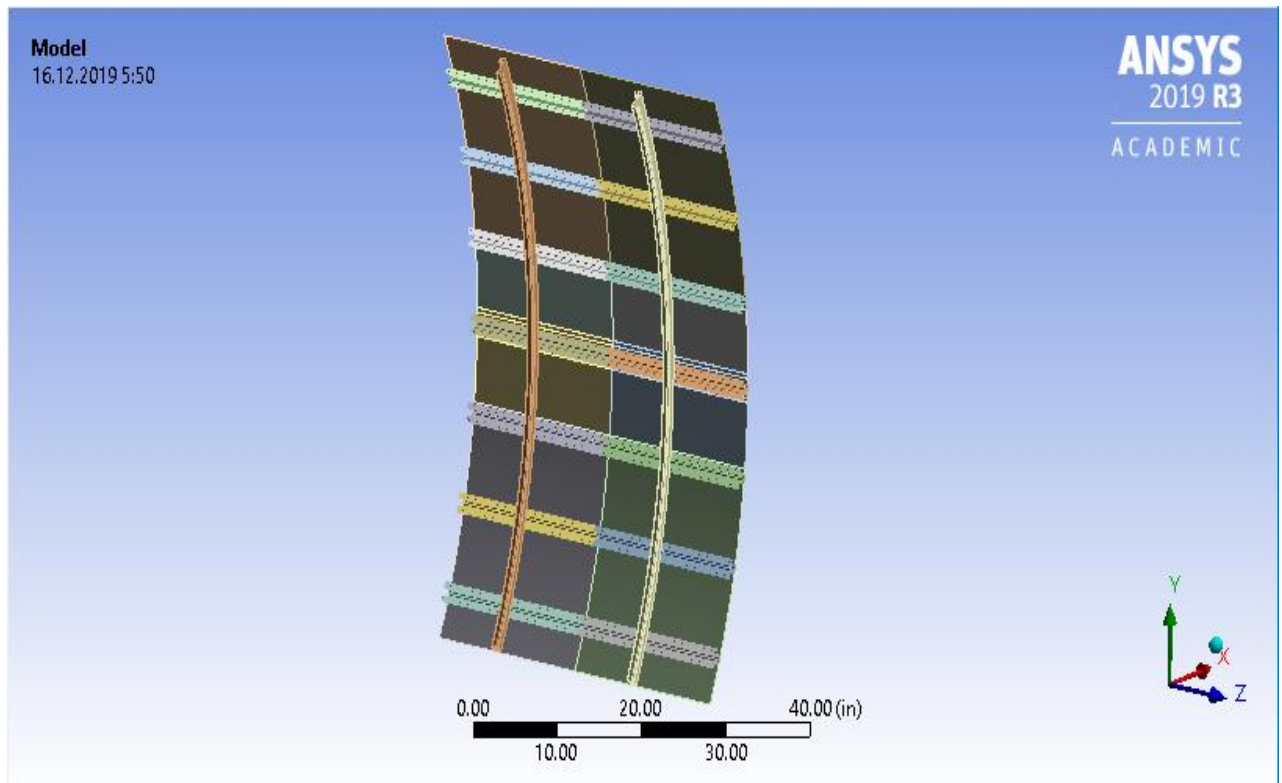


Fig. 3.25 – Geometrical model of 20-inch frame spacing

The analysis model is created and prepared using SpaceClaim (ANSYS2019R3 Academic).

All structural parts are modelled as shells. It is acceptable due to small thicknesses and loading. It is modelled one bay between two frames and a half of bay from both sides of frames.

All dimensions and properties are fixed during analysis except of frame cross section thickness. Chancing of skin thickness – see Fig. 3.26.

Other properties, loads, boundary conditions and mesfing approaches are tha same as used for study of the influence of frame layout on the distribution of forces in the skin longitudinal lap splice, see above for details.

▢	Graphics Properties	
▢	Definition	
	Suppressed	No
	Dimension	3D
	Stiffness Behavior	Flexible
	Coordinate System	Default Coordinate Sys...
	Reference Temperature	By Environment
	Thickness	5.e-002 in
	Thickness Mode	Manual
	Offset Type	Middle
	Treatment	None
▢	Material	
	Assignment	Aluminum Alloy
	Nonlinear Effects	Yes
	Thermal Strain Effects	Yes

Fig. 3.26 – Shell thickness of frame cross section

4. OVERVIEW RESULTS

BEAM188 – Fasteners loads output

It is necessary to correct output results of analysis. Due to this let's refer to Ansys help:

BEAM188 Output Data

The solution output associated with these elements is in two forms: Nodal displacements and reactions included in the overall nodal solution. Additional element output as described in Table 188.1: BEAM188 Element Output Definitions [2].

To view 3-D deformed shapes for BEAM188, issue an OUTRES,MISC or OUTRES,ALL command for static or transient analyses. To view 3-D mode shapes for a modal or eigenvalue buckling analysis, you must expand the modes with element results calculation active (via the MXPAND command's Elcalc = YES option) Definitions [2].

Linearized Stress:

It is customary in beam design to employ components of axial stress that contribute to axial loads and bending in each direction separately; therefore, BEAM188 provides a linearized stress output as part of its SMISC output record, as indicated in the following definitions [2]:

SDIR is the stress component due to axial load:

$SDIR = F_x/A$, where F_x is the axial load (SMISC quantities 1 and 14) and A is the area of the cross-section [2].

SByT and SByB are bending-stress components:

$$SByT = -M_z * y_{max} / I_{zz}$$

$$SByB = -M_z * y_{min} / I_{zz}$$

$$SBzT = M_y * z_{max} / I_{yy}$$

$$SBzB = M_y * z_{min} / I_{yy}$$

where M_y , M_z are bending moments in the beam coordinate system (SMISC quantities 2,15,3,16), as shown in Figure 188.1: BEAM188 Geometry. Coordinates y_{max} , y_{min} , z_{max} , and z_{min} are the maximum and minimum y, z coordinates in the cross-section measured from the centroid. Values I_{yy} and I_{zz} are moments of inertia of the cross-section. Except for the ASEC type of beam cross-section, the program uses the maximum and minimum cross-section dimensions. For the ASEC type of cross-section, the maximum and minimum in each of y and y direction is assumed to be +0.5 to -0.5, respectively [2].

Corresponding definitions for the component strains are:

$$EPELDIR = E_x$$

$$EPELByT = -K_z * y_{max}$$

$$EPELByB = -K_z * y_{min}$$

$$EPELBzT = K_y * z_{max}$$

$$EPELBzB = K_y * z_{min}$$

where E_x , K_y , and K_z are generalized strains and curvatures (SMISC quantities 7,8,9, 20,21 and 22) [2].

The reported linearized stresses are strictly valid only for elastic behavior of members. BEAM188 always employs combined stresses in order to support nonlinear material behavior. When the elements are associated with nonlinear materials, the component stresses can at best be regarded as linearized approximations and should be interpreted with caution [2].

When using KEYOPT(7) with the cubic option (KEYOPT(3) = 3), the integration point at the middle of the element is reported last in the integration-point printout.

The Element Output Definitions table uses the following notation:

In the table below, the O column indicates the availability of the items in the file Jobname.OUT. The R column indicates the availability of the items in the results file [2].

In either the O or R columns, “Y” indicates that the item is always available, a number refers to a table footnote that describes when the item is conditionally available, and “-” indicates that the item is not available [2].

According to above information, the commands for fastener loads output is created, see Fig. 4.1 for details.

```

Commands
1  ! Commands inserted into this file will be executed immediately after the ANSYS /POST1 command.
2
3  ! Active UNIT system in Workbench when this object was created: U.S. Customary (in, lbm, lbf, s, V, A)
4  ! NOTE: Any data that requires units (such as mass) is assumed to be in the consistent solver unit system.
5  ! See Solving Units in the help system for more information.
6
7  /POST1
8  SET, LAST
9
10 ESEL,S,ENAME,,100
11 ALLSEL,BELOW,ELEM
12 NSEL,R,LOC,Y,-1.1,1.1
13 CSYS, 1
14 SELTOL, 1e-3
15 NSEL,R,LOC,X,74.04,74.04
16 CM,N_rivets,NODE
17
18 *GET, n_r, NODE, 0, COUNT
19 n_r = n_r/3
20 *DIM, res, ARRAY, n_r, 4
21
22 CSYS, 0
23 *GET, zmax, NODE, 0, MXLOC, Z
24 *GET, zmin, NODE, 0, MNLOC, Z
25 *GET, ymax, NODE, 0, MXLOC, Y
26 *GET, ymin, NODE, 0, MNLOC, Y
27
28
29 *DO, i, 1, n_r
30   z_i = zmin + ((i-1)/(n_r-1))*(zmax-zmin)
31   res(i,1) = z_i
32   ! upper row
33   CMSEL, S, N_rivets
34   NSEL, R, LOC, Z, z_i, z_i
35   NSEL, R, LOC, Y, ymax, ymax
36   ESLN, S
37   ESEL, R, ENAME, , 100
38   i_e = ELNEXT(-1)
39   *GET, s6, ELEM, i_e, SMISC, 6
40   *GET, s5, ELEM, i_e, SMISC, 5
41   res(i,2) = max(s5,s6)
42   ! middle row
43   CMSEL, S, N_rivets
44   NSEL, R, LOC, Z, z_i, z_i
45   NSEL, R, LOC, Y, 0, 0
46   ESLN, S
47   ESEL, R, ENAME, , 100
48   i_e = ELNEXT(-1)
49   *GET, s6, ELEM, i_e, SMISC, 6
50   *GET, s5, ELEM, i_e, SMISC, 5
51   res(i,3) = max(s5,s6)
52   ! lower row
53   CMSEL, S, N_rivets
54   NSEL, R, LOC, Z, z_i, z_i
55   NSEL, R, LOC, Y, ymin, ymin
56   ESLN, S
57   ESEL, R, ENAME, , 100
58   i_e = ELNEXT(-1)
59   *GET, s6, ELEM, i_e, SMISC, 6
60   *GET, s5, ELEM, i_e, SMISC, 5
61   res(i,4) = max(s5,s6)
62 *ENDDO
63
64 *MWRITE, res, 'forces', 'csv'
65 (F6.2,',',',',3(F12.4,',','))

```

Fig. 4.1 – Commands for fastener loads output

Results of analysis – fastener loads in lap splice for 6 cases of frame layout

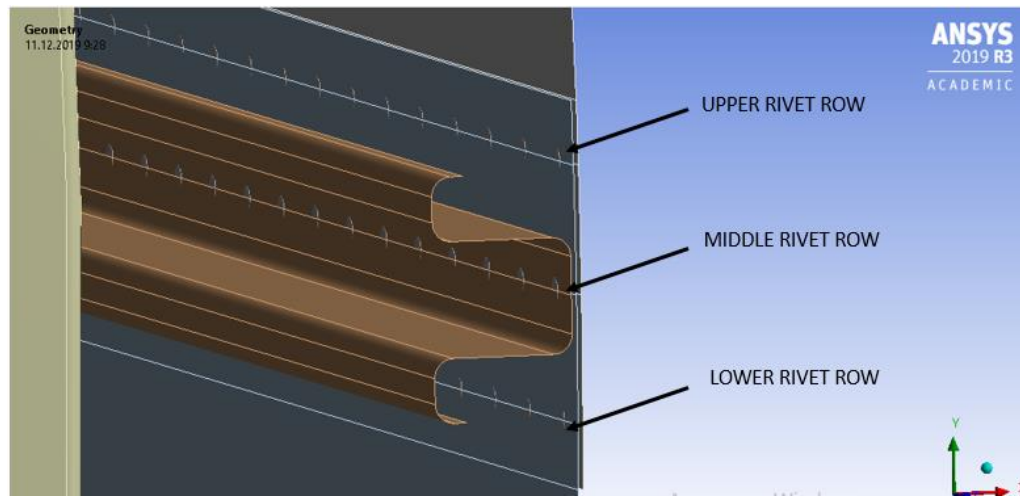
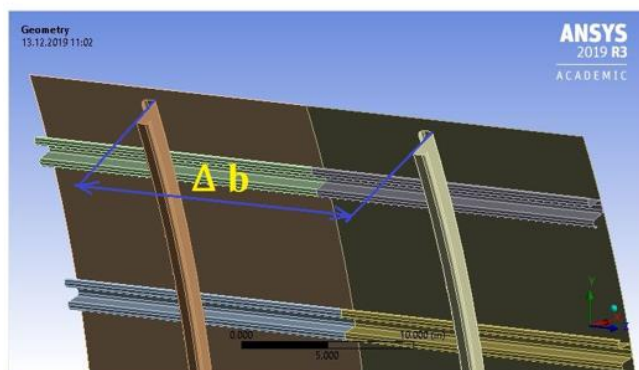


Fig. 4.2 – Rivet rows locations



- Distance between frames Δb increased from 508 mm (20 inches) to 762 mm (30 inches) with 50.8 mm (2.0 inches) increment

Fig. 4.3 – Distance between frames

- **Upper row:** load increased on **4.2%** in comparison of 20 in and 30 in bays (+1.03% → +0.93% → +0.79% → +0.72% → +0.62 %)
- **Middle row:** load increased on **0.63%** in comparison of 20 in and 30 in bays (+0.28% → +0.18% → +0.11% → +0.05% → +0.01 %)
- **Lower row:** load increased on **13.82%** in comparison 20 in and 30 in bays (+3.7% → +3.2% → +2.7% → +2.3% → +1.92 %)

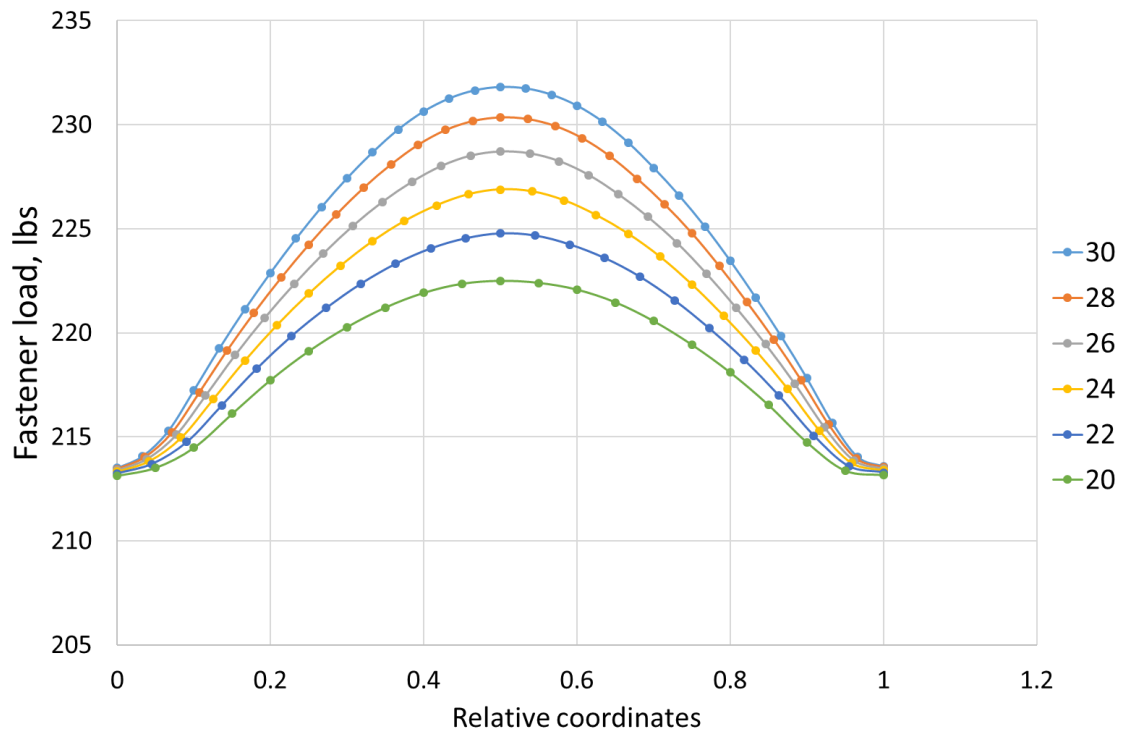


Fig. 4.4 – Upper rivet row (4.2% of load difference between 20 in and 30 in bays)

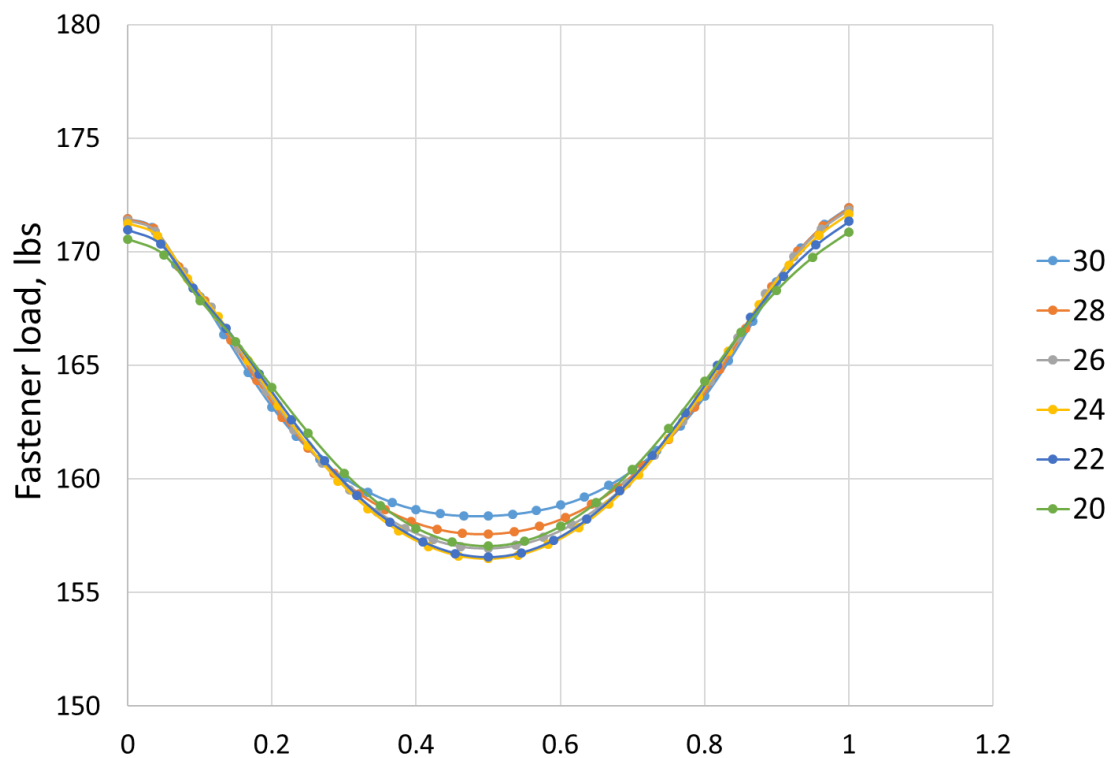


Fig. 4.5 – Middle rivet row (0.63 % of load difference between 20 in and 30 in bays)

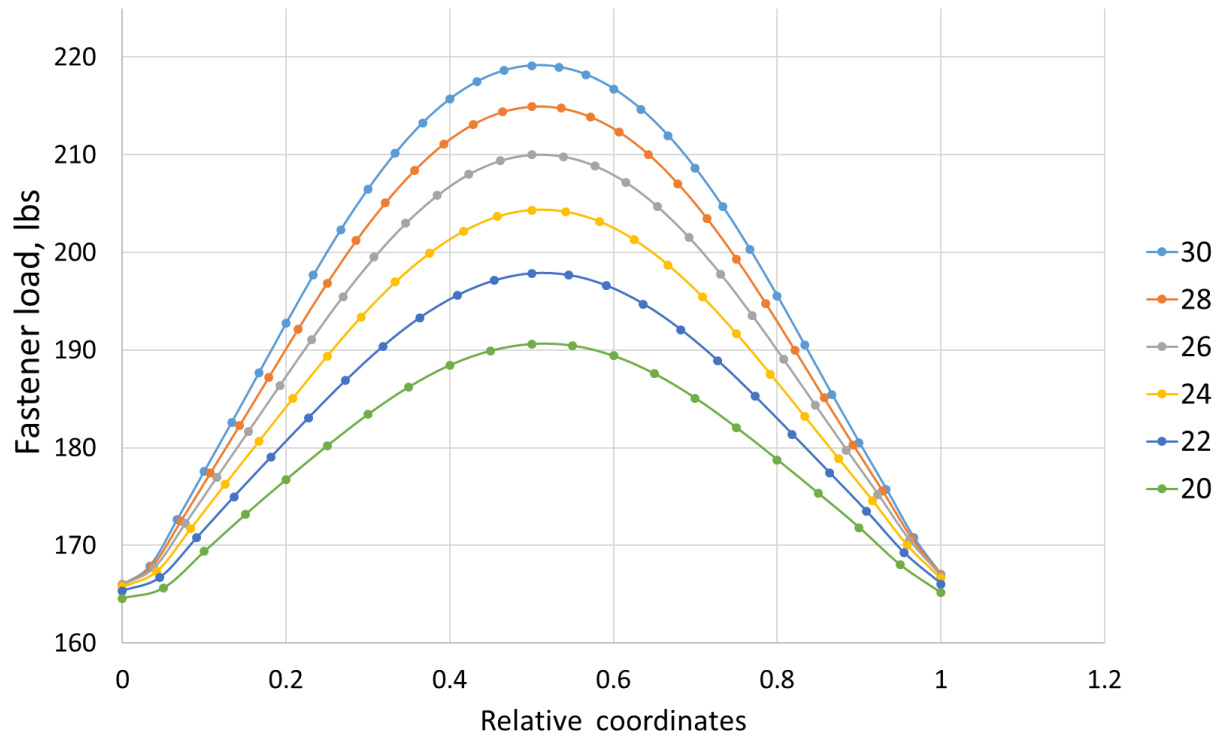


Fig. 4.6 – Lower rivet row (13.82 % of load difference between 20 in and 30 in bays)

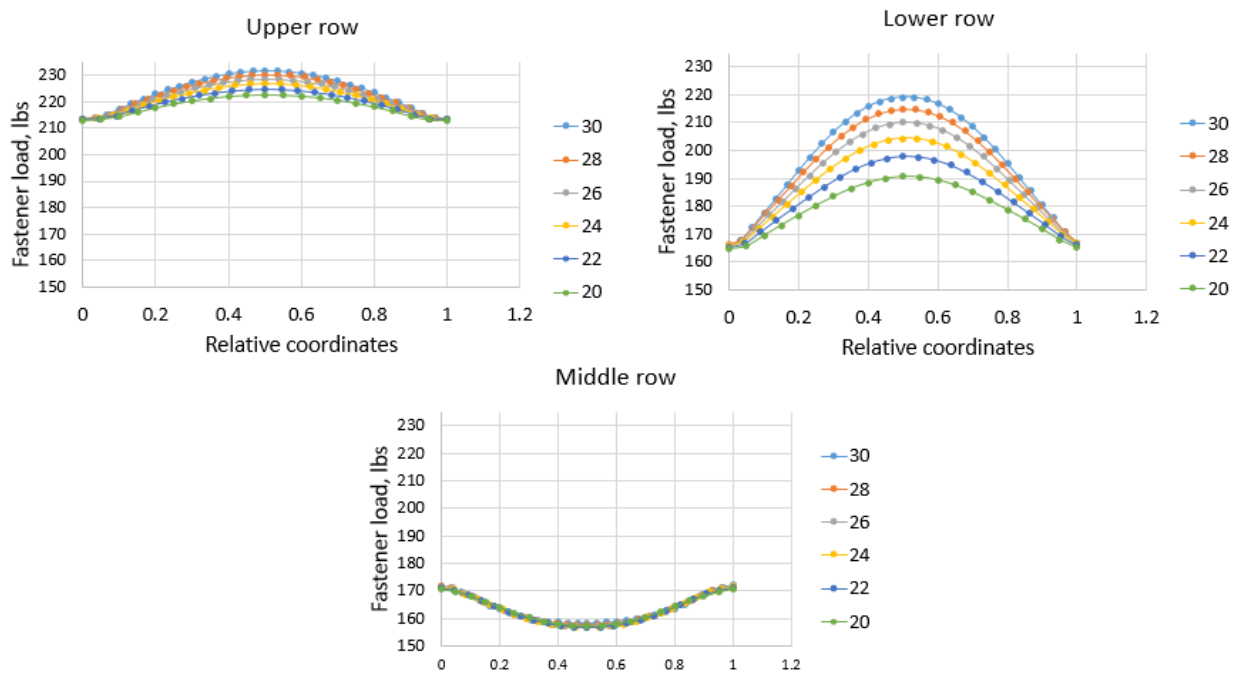


Fig. 4.7 – Comparison fastener loads for all rows and analysis cases

Results of analysis – fastener loads in lap splice for 6 cases of frame rigidly

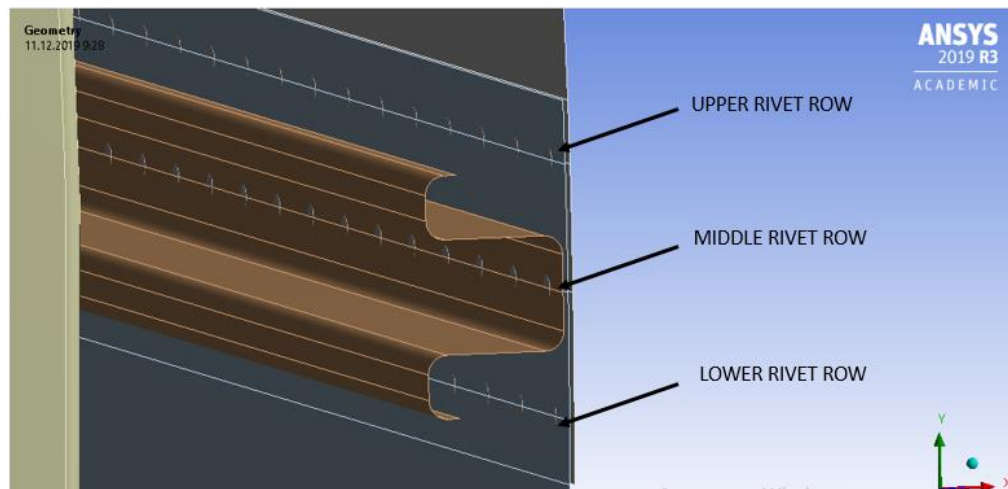
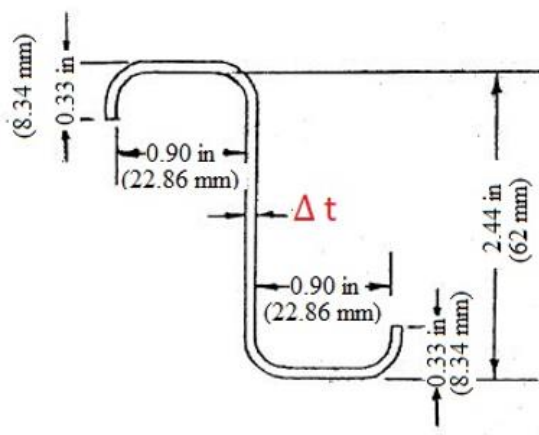


Fig. 4.8 – Rivet rows locations



- Frame thickness Δt is increased from 1.27 mm (0.05 inches) to 1.91 mm (0.075 inches) with 0.127 mm (0.005 inches) increment

Fig. 4.9 – Cross section of frame

- **Upper row:** load decreased on 0.97% in comparison of 0.05 in thick and 0.075 in thick frames (-0.28% → -0.23% → -0.19% → -0.15% → -0.12 %)
- **Middle row:** load decreased on 3.25% in comparison of 0.05 in thick and 0.075 in thick frames (-0.7 % → -0.66% → -0.65% → -0.63% → -0.61%)
- **Lower row:** load decreased on 7.93% in comparison of 0.05 in thick and 0.075 in thick frames (-1.75% → -1.66% → -1.58% → -1.51% → -1.43 %)

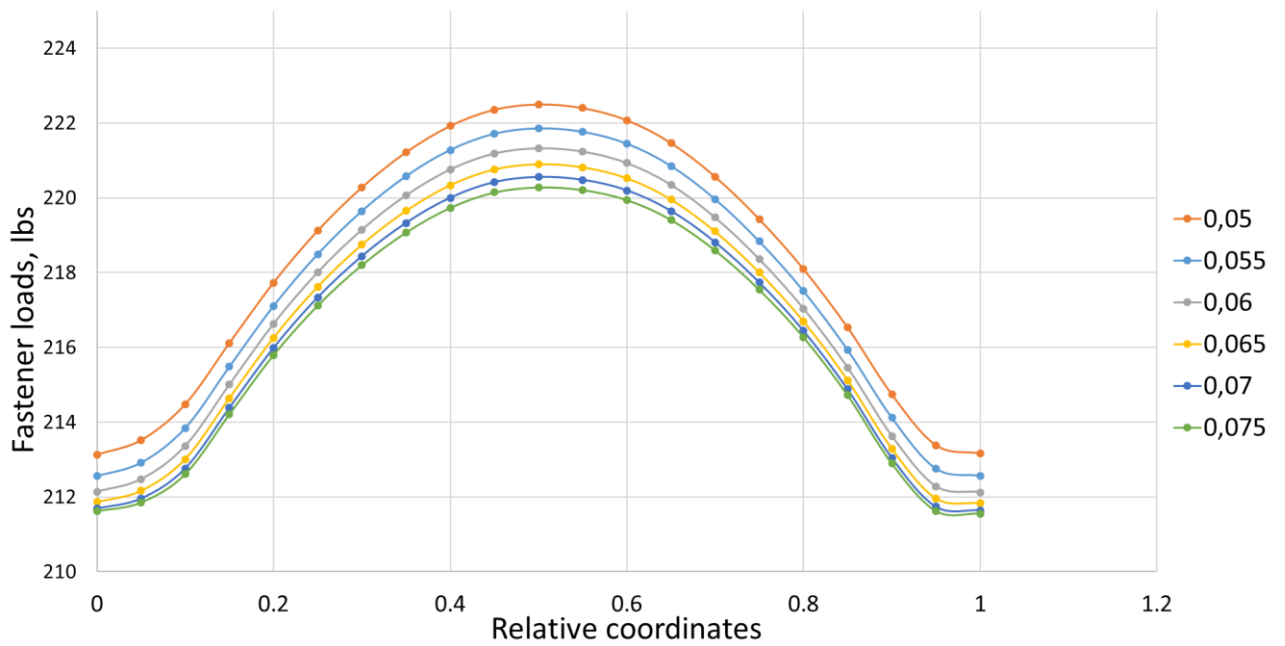


Fig. 4.10 – Upper rivet row (0.97% of load difference between 0.05 in thick and 0.075 in thick frames)

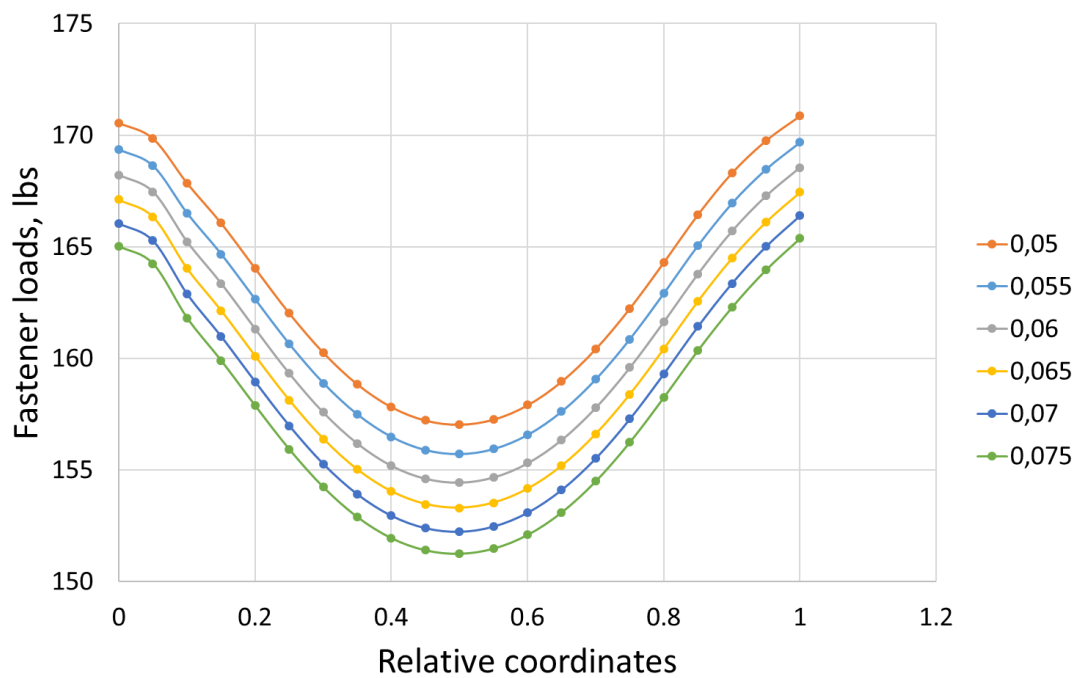


Fig. 4.11 – Middle rivet row (3.25% of load difference between 0.05 in thick and 0.075 in thick frames)

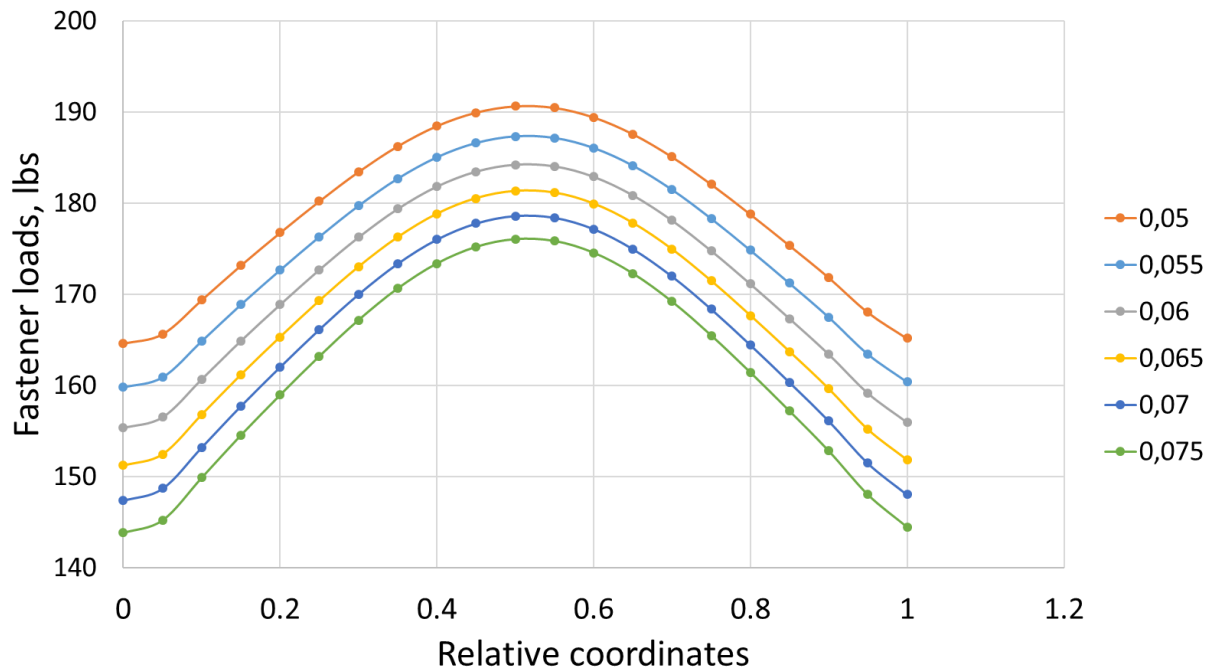


Fig. 4.12 – Lower rivet row (7.93% of load difference between 0.05 in thick and 0.075 in thick frames)

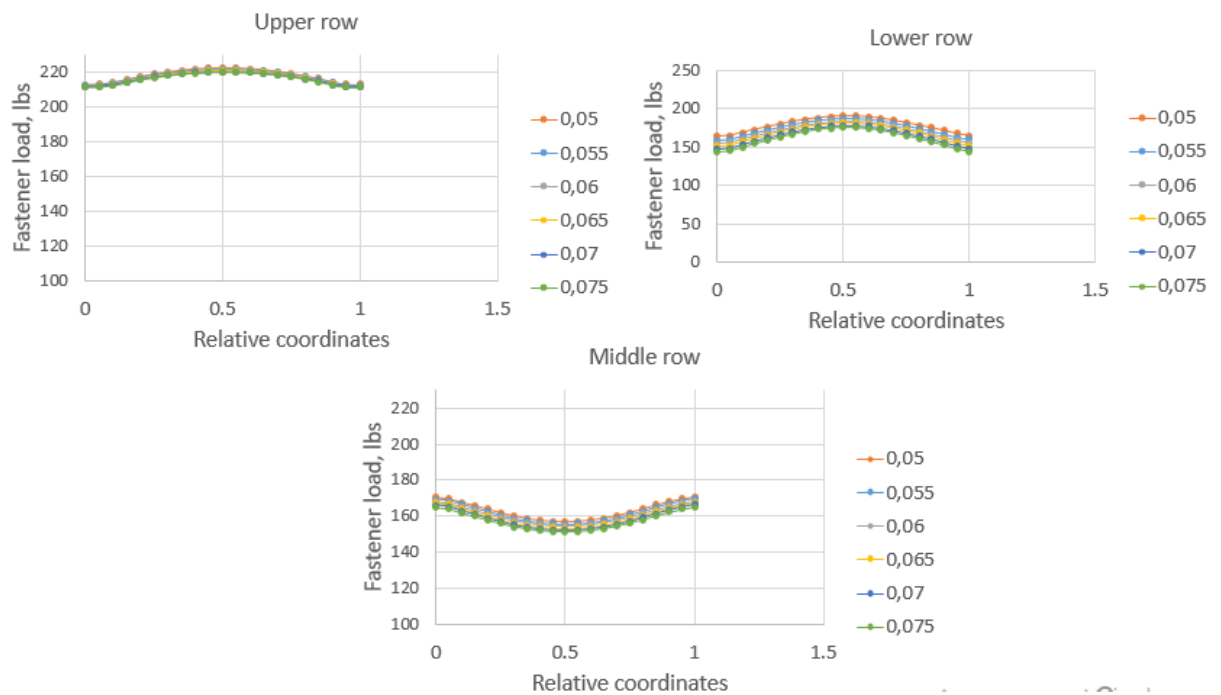


Fig. 4.13 – Comparison fastener loads for all rows and analysis cases

5. STARTUP DEVELOPMING

Description of the project

A marketing analysis of the start-up project was carried out, as well as the importance of the ability to market the market.

Table 5.1

Description of a startup project

Project Summary	Directions of application	Benefits for the user
Using a method to evaluate the effect of rigidity and layout of the fuselage frames on the distribution of forces in the skin longitudinal lap joint	Aircraft structure developmng and optimization	1) Optimization of aircraft weight
		2) Developmng more economical aircraft

The method proposed by the author of the influence of rigidity and layout of the fuselage frames on the distribution of forces in the skin longitudinal lap joint gives rise to the development of this direction, because in Ukraine there are no such experiments. Competitors in our work are: foreign colleagues in the field of aviation.

Technology audit

It is possible to realize the idea of the project due to using FEM.

In the table. 5.2 the analysis of the potential technical and economic advantages of this idea compared to competitor № 1 (foreign colleagues in the field of aviation).

Table 5.2

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

№ п/п	Technical and economic characteristics of the idea	W	N	S
1.	Cash expenses	Competitor 1 №	—	My project
2.	Method of evaluation	—	Competitor №1	My project
3.	The complexity of the manufacture	—	—	—

Determination of technological feasibility of the project idea is given in table. 5.3.

Table 5.3

Technological feasibility of the project idea

№ п/п	The idea of the project	Its technology implementation	The presence of technology	Technology availability
1.	Using a method to evaluate the effect of rigidity and layout of the fuselage frames on the distribution of forces in the skin longitudinal lap joint	FEM analysis	There is	There is
		Practical use		
The selected technology can be implemented				

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

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 our project. To begin with, we will conduct a demand analysis: demand availability, volume and dynamics of market development (Table 5.4)

Table 5.4

Preliminary description of a potential startup project market

№ п/п	Market state indicators	Characteristic
1.	Number of main players, units	1
2.	Total sales, UAH / unit	100
3.	Market dynamics	grows
4.	Sign-in restrictions	not available
5.	Specific requirements for standardization and certification	available
6.	The average rate of profitability in the industry, %	100%

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

Identify potential customer groups.

Potential customer groups can be roughly divided into primary and secondary customers. The primary group is the “Antonov” factory. In the future, we will identify potential customer groups (Table 5.5).

Table 5.5

Characteristics of potential clients of a startup project

№ п/п	The need that shapes the market	Target audience	Differences in the behavior of different potential target customers	Consumer requirements for the product
1.	Developing and optimization of aircraft structure	Airplane factory	finances	Weight optimization

After identifying potential customer groups, we analyze the market environment, that is, compile a table of factors that contribute to market implementation of the project and the factors that impede it (Table 5.6)

Table 5.6

Threat factors

№ п/п	Factor	Contents of the threat	The reaction of the company is possible
1.	Load distribution on lap splice	Increased of load	Improvement of already existing methodology
2.	Development of an improved method of assessment method		

Table 5.7

Opportunity factors

№ п/п	Factor	Content features	The reaction of the company is possible
1.	It does not require big expenses	Frugality	Decent pay
2.	Increased demand in the industry	Increase in application	Weight optimization and lower cost

In the following we will analyze the proposal, defining the general features of competition in the market (table 5.8).

Table 5.8

Step-by-step analysis of competition in the market

Features of the competitive environment	In what this characteristic is manifested	Impact on the activity of the enterprise
Type of competition aviation	A small number of sub- enterprises, which are mostly non-price competitors.	Not noticeable impact
National level of competition	This market is quite open	Significant impact on the modernization of the project
Intra-industry trait	The struggle is being fought between experimental methods	Savings due to the introduction of the methodology
Commodity competition by type of goods	Ability to meet industry needs	Not a lot of competitors
Non-price competition	Improving the quality of the technique compared to others	Work on continuous improvement of the methodology
Vintage intensity	We consider similar methods as competitors	Independence from the market

After the analysis of competition, we conduct a more detailed analysis of the conditions of competition in the industry (table 5.9)

Table 5.9

Analysis of competition in the industry by M. Porter

Components of analysis	Direct competitors in the industry	Potential competitors	Customers	Substitute goods
	None	There are no barriers to entry	High impact factor	There is no threat from substitutes
Conclusions :	Low intensity	It is possible to enter the market	Market conditions dictate. Such as time to calculate and provide the right software	No restrictions on the impact of substitutes

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 in table 5.9, and taking into account the characteristics of the idea of the project table 5.2, consumer requirements for the table table 5.5 and factors of the marketing environment table 5.6-5.7 determine and justify the list of factors of competitiveness. The analysis is formalized in table 5.10.

Table 5.10

Justification of the factors of competitiveness

№ п/п	Competitiveness factor	Rationale (citing factors that make the comparison of competing projects meaningful)
1	There is less need for costs	No need to-secondary operations
2	Test accuracy	Improving results
3	Lost cost	Maximum resource depletion

According to the identified factors of competitiveness (Table 5.10) we will analyze the strengths and weaknesses of my startup project (Table 5.11).

Table 5.11

Comparative analysis of strengths and weaknesses «Influence of rigidity and layout of the fuselage frames on the distribution of forces in the skin longitudinal lap joint»

№ п/п	Competitiveness factor	Points 1-20	Rating of competing products in comparison with the project « <i>Influence of rigidity and layout of the fuselage frames on the distribution of forces in the skin longitudinal lap joint</i> »						
			-3	-2	-1	0	1	2	3
1	There is 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 final stage of market analysis of project implementation opportunities is the compilation of SWOT analysis (Strength and Weak matrix, Troubles and

Opportunities) (Table 5.12) on the basis of selected market threats and opportunities, and strengths and weaknesses (Table 5.11).

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 on the market and are likely to occur.

Table 5.12

SWOT- analysis of a startup project

Strengths: airplane cost may decrease	Weaknesses: Not perfectly accurate results usng FEM
Opportunities: conquering the entire market	Threats: Develop a more accurate methodology.

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 obtaining resources (Table 5.13).

Table 5.13

Alternatives to market introduction of a startup project

№ п/п	An alternative to market behavior	The probability of receiving resources	Terms of implementation
1	Public review, review of existing studies (analogues), state approval	High	2 months
2	Public review, validation on a real experiment, state approval	High	10 months

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.

Develop a market strategy for the project

Developing a market strategy as a first step involves defining a market outreach strategy: a description of the target groups of potential consumers (Table 5.14).

Table 5.14

Selection of target groups of potential consumers

№ п/п	Description of the profile of your target audience	Willingness of consumers to accept the product	Targeted demand within the target group	Intensity of competition in the segment	Easy entry to the segment
1	“Antonov” factory	Completely ready	High	Low	Easy

To work in the selected market segment, it is necessary to form a basic development strategy (Table 5.15).

Table 5.15

Defining a basic development strategy

Selected alternative to the project development	Market outreach strategy	Key competitive positions according to the alternative chosen	Basic development strategy
Alternative	80-90% coverage	competitor.№1	Specialization

Next, choose a strategy for competitive behavior (table 5.16).

Table 5.16

Defining a basic strategy for competitive behavior

Is the project a pioneer in the market?	Will the company look for new customers or take on existing competitors?	Will the company copy the key features of a competitor's to-bar, and which ones?	Competitive behavior strategy
Yes	No	No	Industry occupation strategy

- The result of this section is a coordinated system of decisions regarding the market behavior of a startup company, which will determine the directions of work of a startup company in the market:
 - - work with existing customers;
 - - use of modern technologies in the project;
 - - the direction of work only in a given niche of the market.

Develop a marketing program for a startup project

The first step is to formulate a marketing concept for the product that the consumer will receive. For this purpose in table. 5.17 summarize the results of the preliminary analysis of the competitiveness of the goods.

Table 5.17

Identify the key benefits of the concept of a potential product

№ п/п	Need	The benefit that the product offers	Key advantages over competitors
1	It does not require large expenses for calculations	Зменшення собівартості	Decent pay
2	Increased demand in the field of medicine	Reduction of operations	Increasing the speed of rehabilitation

Next, we determine the optimal sales system within which the decision is made (Table 5.18).

Table 5.18

Formation of sales system

Specificity of purchase behavior of target customers	Sales functions to be performed by the vendor	Depth of sales channel	Optimal sales system
One-time purchase	Informing	Deep	State approval of the methodology

The last component of the marketing program is the development of the concept of marketing communications, which relies on a pre-selected basis for positioning, certain specific behavior of customers (table 5.19).

Table 5.19

Marketing communications concept

Specific behavior of target customers	Communication channels used by target customers	Key positions selected for positioning	The task of the advertising message	Advertising concept
Communicative	Internet, scientific works	Advertising on the Internet and conferences	Target customers extension	Show that our product is the best

Marketing research results:

- my startup project is up-to-date because it can take developing to the next level;
- the uniqueness of the idea;
- market introduction is appropriate and can generate revenue, especially if introduced into the aviation field.

6. CONCLUSIONS

1. The method was developed and implemented for study the influence of rigidity and layout of the fuselage frames on the distribution of forces in the skin longitudinal lap joint.

2. Based on the obtained results the dependences between the distribution of efforts in the longitudinal joint and the the layout frames and their rigidity are established.

3. During pressurization, the mid bay of skin (midway between stringers) pillows due to the pressure but is held at centerline of stringers due to the stringer stiffness, but the stringer is free to rotate. The stringers are restrained from rotating at the frames. Mid bay rotation is a function of stringer torsional stiffness. This adds a twisting moment at the lap splice, causing additional local bending in addition to the hoop pressure load. The applied moment is partly reacted by the moment due to the offset skin hoop loads. Increased skin flexibility increases joint rotation, and the moment carried by the offset skin hoop loads.

4. Increasing of the frame spacing increases the uneven nature of the distribution of forces in the joints. The frame rigidly affects the distribution of efforts only quantitatively, but not qualitatively.

5. Obtained data is validated by Boeing analysis methods and approaches that are Boeing proprietary source.

6. This analysis method can be used during developping and optimization of aircraft structure.

REFERENCE LIST

1. Michael C.Y Niu (March, 1988)“Airframe structural design”(pp. 1 – 612).
U.S.A. California
2. Ansys Help Manual for ANSYS2019R3 Academic,
<https://ansyshelp.ansys.com/> (An electronic source)