STABILITY ANALYSIS OF ADDITIVELY MANUFACTURED ISOGRID

Thesis

Submitted to

The School of Engineering of the

UNIVERSITY OF DAYTON

In Partial Fulfillment of the Requirements for

The Degree of

Master of Science in Engineering Mechanics

By

Sirija Ananth

UNIVERSITY OF DAYTON

Dayton, Ohio

December, 2015
STABILITY ANALYSIS OF ADDITIVELY MANUFACTURED ISOGRID

Name: Ananth, Sirija

APPROVED BY:

__________________________________________________________________________
Thomas J Whitney, Ph.D.
Advisory Committee Chairman
Assistant Professor
Civil and Environmental Engineering
and Engineering Mechanics

Elia Toubia, Ph.D.
Committee Member
Assistant Professor
Civil and Environmental Engineering
and Engineering Mechanics

__________________________________________________________________________
Steven Donaldson, Ph.D.
Committee Member
Assistant Professor
Civil and Environmental Engineering
and Engineering Mechanics

__________________________________________________________________________
John G. Weber, Ph.D.
Associate Dean
School Of Engineering

Eddy M. Rojas, Ph.D., M.A., P.E.
Dean,
School of Engineering
ABSTRACT

STABILITY ANALYSIS OF ADDITIVELY MANUFACTURED ISOGRID

Name: Ananth, Sirija
University of Dayton
Advisor: Thomas J Whitney, Ph.D.

This work investigated the stability of an isogrid panel manufactured using fused deposition modelling (FDM) In particular, it verifies the use of existing closed form analytical solutions and finite element analyses for predicting both global and local buckling loads and modes for these structures FDM-produced isogrid samples were subjected to uniaxial quasi-static compression with boundary conditions approximating simple supports. Buckling values and mode shapes were obtained from Digital Image Correlation (DIC). The values obtained experimentally were compared to buckling loads calculated using finite element analysis and closed form solutions for orthotropic materials. Excellent results were obtained in comparing finite element analysis to experimental results for both global and local modes, while closed-form solutions compared well for the global modes for which the solutions were intended.
Dedicated to my parents
ACKNOWLEDGMENTS

I would like to express my sincere gratitude to Dr. Thomas J Whitney for being my advisor, for providing the time and trusting in me, for motivating me from time to time, and. I would like to thank Dr. Elias Toubia for spending his valuable time in my research. I would like thank both of them for directing me through my thesis and help me to obtain results.

I would like thanks Dr. Li Cao for helping me to prepare my specimen. I would like to thank Susan Hill for helping me with laboratory and result analysis.

I also place on record, my sense of gratitude to one and all, who directly or indirectly, have lent their hand in this venture.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ABSTRACT</td>
<td>vi</td>
</tr>
<tr>
<td>DEDICATION</td>
<td>v</td>
</tr>
<tr>
<td>ACKNOWLEDGMENTS</td>
<td>vi</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>ix</td>
</tr>
<tr>
<td>LIST OF TABLES</td>
<td>xi</td>
</tr>
<tr>
<td>LIST OF ABBREVIATIONS AND NOTATIONS</td>
<td>xii</td>
</tr>
<tr>
<td>CHAPTER 1 INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>CHAPTER 2 REVIEW OF RELATED LITERATURE</td>
<td>4</td>
</tr>
<tr>
<td>CHAPTER 3 MANUFACTURING</td>
<td>10</td>
</tr>
<tr>
<td>DIC Testing Conditions</td>
<td>20</td>
</tr>
<tr>
<td>Finite Element Analysis Procedure</td>
<td>26</td>
</tr>
<tr>
<td>Closed-Form Approach</td>
<td>32</td>
</tr>
</tbody>
</table>
CHAPTER 4  RESULTS AND DISCUSSION........................................................................... 35

Specimen S1.................................................................................................................. 35
Specimen S2.................................................................................................................. 46
Specimen S3.................................................................................................................. 52
Discussion..................................................................................................................... 58

CHAPTER 5  CONCLUSION ......................................................................................... 63

Recommended Future Work:....................................................................................... 65

BIBLIOGRAPHY............................................................................................................. 66
LIST OF FIGURES

1. Isogrid ..................................................................................................................... 2
2. Aluminum Isogrid Curved Panel [22]................................................................. 6
3. FDM and parts [23].......................................................................................... 11
4. Insight interface and sliced layer of Isogrid..................................................... 13
5. Tool path options .............................................................................................. 14
6. Fortus 400mc machine specifications [21]...................................................... 15
7. Mechanical and Thermal Properties of the ULTEM™ 9085[20].............. 17
8. Dimensions of Specimen S1 ........................................................................ 18
9. Dimensions of Specimen S3 ........................................................................ 19
10. DIC experimental setup .................................................................................. 21
11. Specimen S1 and S2 ...................................................................................... 22
12. Specimen S3 ................................................................................................... 22
13. DIC experimental setup 2 .............................................................................. 24
14. Loading directions and boundary conditions on S1 and S2........................ 25
15. Meshed Isogrid ............................................................................................. 28
16. Boundary conditions in FEA ......................................................................... 29
17. Loading and boundary conditions for Specimen S1 and S2 ................................. 30
18. Loading and boundary conditions for Specimen S3 ............................................. 31
19. Grid terminology .................................................................................................. 33
20. Step vs Load graph Specimen S1 ........................................................................ 37
21. Step vs displacement in ‘z’ graph Specimen S1 .................................................. 37
22. Displacement of Specimen S1 in ‘z’ ..................................................................... 38
23. 1-4 buckling modes of Specimen S1 .................................................................... 42
24. Step vs Load graph Specimen S2 ........................................................................ 47
25. Step vs Displacement in ‘z’ graph Specimen S2 .................................................. 47
26. Displacement of Specimen S2 in ‘z’ ..................................................................... 48
27. 1-4 buckling modes of specimen S2 ..................................................................... 51
28. Step vs Load Specimen S3 .................................................................................... 53
29. Step vs Displacement in ‘z’ direction Specimen S3 ............................................. 53
30. Displacement of Specimen S3 in ‘z’ ..................................................................... 54
31. 1 – 4 buckling modes for specimen S3 ................................................................. 57
32. First buckling modes from DIC and FEA Specimen S1 ....................................... 60
33. First buckling modes from DIC and FEA Specimen S2 ....................................... 60
34. First buckling modes from DIC and FEA Specimen S3 ....................................... 61
35. Rib Failure Specimen S2 ...................................................................................... 61
LIST OF TABLES

1. FEA buckling values for specimen S1 ................................................................. 39
2. Material properties for all Specimen S1 S2 and S3 ......................................... 43
3. FEA buckling values for Specimen S2 ................................................................. 49
4. FEA buckling values for Specimen S3 ................................................................. 55
5. Critical buckling loads of all three specimen ..................................................... 58
6. Percentage error of FEA and Closed form approach to DIC results ............... 62
LIST OF ABBREVIATIONS AND NOTATIONS

$E_1$ Young’s modulus along $x$

$E_2$ Young’s modulus along $y$

$E_3$ Young’s modulus along $z$

$\nu$ poisons ratio

$S_1, S_2, S_3$ Specimen 1, 2 and 3 respectively

$P_x$ Critical load

$m, n$ Eigen modes

$a, b$ length and width of the plate

$t$ thickness
CHAPTER 1

INTRODUCTION

The usage of Isogrid structures in aerospace, marine, automobile and mechanical components is widening due to exceptional stiffness to weight ratios. Fuselage and fuel tanks are successful applications of such structures. The increasing demand of composite materials in these industries has significantly increased their performances. In addition to structural efficiency and materials, efficient and reliable manufacturing procedures have also impacted the final design.

Grid structures have been in use since World-War II. Most of them were manufactured using metals and concrete. Composite grids offer high resistance to loads and provide high stiffness to low mass. Grids are different from sandwich constructions. Grid structures can behave like isotropic material due to the internal ribs orientation. In these structures, grids are the primary load carrying members. The skin is used to cover the ribs and transfer loads through membrane action. Types of isogrid and geometry of its structure is shown below in Figure 1.
The 3D printing technology is an emerging manufacturing procedure, also called “desktop fabrication” or “additive manufacturing”. A real object is manufactured using digital object designed using any CAD software (AutoCAD or SolidWorks [19]). 3D printing adopts different methods like selective laser sintering (SLS), Fused Deposition Modeling (FDM), and Stereolithography (SLA) [1]. This technology has evolved to the point that one can print anything if the object can be drawn.

The ULTEM™ 9085 resin (a blend of polyetherimide and polycarbonate resins) is well known in aerospace and marine industries because of its high strength to weight ratio and its FST (flame, smoke and toxicity) rating. This material can be printed to produce a functional part.

The DIC (Digital Image correlation) technique used to validate the FEA and analytical results is a full-field 3D technique to measure contour, vibration or deformation of any material. This method uses non-contact optical data and can be used...
for tensile, torsion bending and any static or dynamic applications. This technique can also be applied to wide range of structures from very small to large testing areas.

**Objective of this study:**

This research work attempts to determine and experimentally validate an existing analytical technique to predict buckling loads of the isogrid structure. This effort will potentially validates its application to the Fused Deposition Modeling manufacturing techniques. Three specimen of different dimensions were manufactured and tested under uniaxial compression. The first two specimens had the same grid dimension, but with different skins thicknesses. Both specimens were tested along the axial stiffeners. A third specimen having the same grid dimensions with thinner skin was subjected to load at a right angle to the axial stiffeners (the grid stiffeners consist of two ribs (stiffeners) which are at ±60° to the long axis of the panel and one rib parallel to the long side of the plate). Since the material used in this research is anisotropic, the “Manufacture and design of composite grids” by Stephen W. Tsai, Kevin K.S. Liu and Philippie M. Manne [15] was used in this research work. This reference presents a detailed method to develop the equivalent young’s modulus for a grid manufactured structure using anisotropic material. This research has proved that grid structures are more efficient than laminates. The analytical steps for grid structure along with skin were Chapter 3.
CHAPTER 2
REVIEW OF RELATED LITERATURE

Usage of metal isogrid dates back to 1950s, mainly in the space program due to their high cost of manufacturing. They were manufactured using milling or assembling of structural ribs to form a grid. Skin evolution started with cloth, then advanced to plywood and then finally to aluminum. Grid structures are usually classified according to their orientation. If each grid is perpendicular to the other they’re called as Ortho Grids. Among the grid family this grid has the lowest performance as they can’t support any other loads apart from those that act in the direction of the grids. Early aircrafts like DC-6 and DC-7 used this type of grids, which were mechanically attached to one another. This type of structures are also used in the Saturn space vehicle in its second stage. Zero degree and 45° ribs were used in its later stages. These grid structures played a prominent role in the early 1970s.

“Anisogrid” patterns usually have a circular form and the grid behave like an anisotropic material. Applications have appeared since 1960, mainly in the Russian heavy space launcher Proton-M.[2]
“Isogrid” has everything in its name, as these structures act like isotropic materials. Isogrid is a structure in which the stiffeners ribs intersect at 60° with each other to form an equilateral grid. When a load is applied to this kind of structure it will be equally distributed among the stiffeners. Figure 2 shows an example of an aluminum isogrid structure.

Isogrid has also a long history, McDonnell Douglas presented many formulas for the composite isogrid in 1973. The report “ISOGRID DESIGN HANDBOOK” [3] has analytical techniques for a wide range of isogrid applications. These technique have been illustrated based on thickness transformation of a unit plate. Using non-dimensional parameters and basic Hooke’s law, Young’s modulus and Poisson’s ratio a grid structure will be transformed into unit width plate values. According to this method a basic grid is transformed into a regular plate with uniform thickness. With known load and boundary conditions one can use many methods to solve various structural problem.

Application and verification of this method is given by Jeffrey Lavin [4], where a simple isogrid was considered. Buckling values obtained from the above method were verified with COMSOL model (a finite element method software package). A few drawbacks of this procedure are also explained in this report.

A list of successful isogrid manufacturing methods, history of isogrid and pros and cons of the structure are is detailed in the paper by Steven M Huybrechts, Steven E. Hahn, Troy E. Meink [5]. The earliest isogrids were made of aluminum and are either milled or assembled.
Composite materials have excellent strength-to-weight and stiffness-to-weight ratios. Directionality of composite materials along the ribs added material strength to grid structure. Grid patterns in a triangular modular behave as an isotropic material giving rise to the name “isogrid”. While manufacturing this grid structure due to overlapping of ribs there’s a possibility of out of plane compaction. Improper fabrication causes higher loads to concentrate at the intersections. Design and buckling loads estimation have been clearly explained in the report “Continuous Filament Advanced Composite Isogrid” by L. W. Rehfield, R. B. Deo [6].

Smooth consistent joints were produced by reworking the silicon modeling procedures used in the mold of a filament winding mandrel for both curved and flat
isogrid. Later, an automated layup machine was invented by James Koury using the filament winding procedure. [7]

When it comes to structures, stability is widely used to define the “strength” of the structure. Buckling is the main failure mode in this context. Study buckling is always difficult, as one has many parameters to consider that affect the buckling. Structural parameters of structure both geometric and material property, load and boundary conditions can change the buckling loads and predictions. Many methods have been introduced to evaluate buckling of grid structure that are still in use real life in practical design. “Optimization of the static and dynamic characteristics of plates with Isogrid stiffeners” by W. Akl, A. El-Sabbagh, and A. Baz uses a finite element model to calculate and optimize the buckling loads and frequencies of the plates and stiffeners.[8]

“Numerical investigation into the buckling behavior of the advanced grid stiffened composite cylindrical shell” by Mingfa Ren, Tong Li, Qizhong Huang and Bo Wang [9] investigated the buckling response of the cylinder using equivalent stiffness model, finite element model and hybrid model combining the first two. Finite element values were compared with the hybrid model values and evaluated independently.

“Numerical Investigation of Post Buckling Strength and Failure Modes in Advanced Grid Stiffened Structure under Thermal-Mechanical Loads” by Ruixiang Bai, Bo Chen, Cheng Yan, Lin Ye, Zecheng Li and Haoran Chen[10] worked on the same cylindrical isogrid but with a different approach of first order deformation theory, and Von Karman non-linear deformation assumptions.

“Fabrication and testing of thin composite isogrid stiffened panel” by Thomas D. Kim, describes the fabrication and axial compression testing of composite isogrid
stiffened panel. This paper identified various failure modes that represent three buckling modes and including skin buckling, rib crippling and general instability.[11]

“Process-induced properties of FDM Products” presents an effective approach for characterization of the evolved mechanical properties of ULTEM™ 9085. This paper used classical lamination theory for composite materials together with experimental procedures, image-based mesostructured and analytical testing techniques for generation of properties. The values from this paper were adopted for this analysis. [12]

“Mechanical Properties of Fused Deposition Modeling parts manufactured with ULTEM™ 9085” by Agnes Bagsik and Volker Schoppner proved that mechanical properties depend on the given inner part structure as a result of build direction and toolpath generation. Methods to obtain best results were explained and proved, like using a negative raster air-gap. Specimen manufactured with conventional inject molding method has a different plastic behavior for the same material.[13]

“Numerical modelling of stresses and buckling loads of isogrid lattice composite structures cylinder” by H Kanou, S M Nabavi, J E Jam studied a numerical model using ANSYS software to find the stress and buckling loads for cylindrical grid structures with and without skin and proved that with increase in the thickness of either skin or grid there will be increase in the buckling load. It was also shown as the skin thickness increases the buckling failure mode of the cylinder shifted from global skin buckling to local buckling and then to stiffener crippling [14]

There are many different analytical approaches used to estimate the buckling value. For example Classic lamination theory (CLPT), the Equivalent stiffness Method,
the Equivalent Thickness Method, the Basic -Cell Energy Equivalence Method, Bloch’s wave method and others have all been used to estimate buckling loads of grid structures.

This research work adopts the necessary formulae to analytically predict the buckling load from “Manufacture and Design of Composite Grids” by Stephen W. Tsai. [15] This Thesis uses the basic terminology of a grid structure using different manufacturing process and mechanical properties of the grids. Effective stiffness and other properties for grid and laminate structures are given. These equations are validated with different examples. This report also discusses and explains different failure modes observed in the DIC technique.
CHAPTER 3
MANUFACTURING

3D Printing:

Additive Manufacturing technologies are mainly divided into seven categories as per “Committee F42, “Additive Manufacturing” of ASTM International [16]. These methods are 1) Vat Photopolymerisation 2) Material Jetting 3) Binder Jetting 4) Material Extrusion 5) Powder Bed Extrusion 6) Sheet Lamination, and 7) Directed Energy Deposition. The most common technologies like Selective Laser Sintering (SLS) and Fused Deposition Modeling (FDM) uses powder bed fusion and material extrusion respectively.

Fused Deposition Modeling (FDM) is an additive manufacturing method that was first invented by Scott Crump in late the 1980’s. He also started Stratasys Company in 1988 to commercialize the technology. Plastic filament is unwound from a coil and supplied to an extrusion nozzle.
This preheated nozzle is capable of melting the material and moving in both horizontal and vertical directions by a numerically controlled mechanism, directly controlled by a computer-aided manufacturing (CAM) software package.

3D Printing or Additive manufacturing is a layering technique. Layers are deposited bottom up by heating and extruding thermoplastic filament. This procedure involves three steps. Pre-processing, Production and Post processing. Pre-processing involves modeling of the part. The 3D model represents a physical body. The model is a mathematical representation of three dimensioned surface of an object using particular software. There are many software packages that can be used for this purpose for example AutoCAD, 3DS MAX, Autodesk Maya, SolidWorks, SketchUp and others. All these software packages enable design of any 3D structure. SolidWorks was used in this research. Once the specimen has been modeled the model uses as a Standard Tessellation Language (STL) file. Figure 3 shows different parts of FDM.

Figure 3 FDM and parts [23]
Pre-processing also involves slicing and defining the tool path. Since additive manufacturing is a layering technology and parts are built bottom-up, each and every 3D model has to be sliced to a minimum thickness. The software used to process the 3D model into layers and define the toolpath is typically specific to the printers used. Since printing in this work was performed in a Stratasys FORTUS 400mc Insight 9.1 [25] developed by Stratasys was used.

Extruded melted material were laid in layers to build the part. Insight enables one to define the path being laid, and direction and thickness of the material. Contours, which are extrusion roads (the term used interchangeably with “toolpaths” and “beads”) laid around the outer boundary of each layer, are also defined automatically in Insight. Certain support structures will also were built to along with the specimen. These shoring members were built in order to support the part. Geometry and orientation of the part plays an important role when software auto generates this support material. This process is a most important step in Additive Manufacturing techniques. Figure 4 shown below are the specific tool paths used to produce the reinforcing grids in the specimens used in this work.
When tool paths are auto generated it is important to verify if each and every part of the specimen has been generated with a tool path. Toolpath properties like usage of contour width, raster angle and width can be controlled as required. Proper use of these options ensure the production of a perfect model. Figure 5 shows the default options for these properties as stored in Insight. The printed specimen were generated using 0.02” raster width and grid tool paths at an angle of ±60º relative along the long edge of the sample. Grids at 90º to this edge were also printed to complete the isogrid Geometry.
Figure 5 Tool path options
Once the 3D model is sliced and toolpaths defined, a “.cmb” file with machine-level instructions for printing is sent to the FORTUS 400mc printer. Specifications for the FORTUS 400mc are shown in Figure 6. The Fortus 400mc coupled with Insight allows the designer to manufacture parts to match the mechanical, thermal and other requirements and specifications. The time to print a specimen depends upon the size and complexity of the model and on the chosen layers thicknesses. If the layers are too thin and numerous, more time will be taken to print the part and consequently leading to a
better surface finish. Post-processing involves removal of the support material using tools and finishing. If the model is built efficiently post processing time is typically minimized.

**Material properties:**

The material used in this research were manufactured using ULTEM™-9085 resin. This resin is based on a polyetherimide/polycarbonate blend is with flame-retardant and high-performance thermoplastic frequently using in additive manufacturing and other applications. Its high FST (flame, smoke and toxicity) rating makes it an excellent choice for the transportation industries like aerospace, marine and ground. Figure 7 lists all the mechanical and thermal properties of this particular material.

ULTEM™ 9085 liquefies at 325 °C, and the build chamber reaches a temperature of 195 °C. Once the material leaves nozzle and deposited, completed parts have a heat deflection temperature of 153 °C. ULTEM™ 9085 has a very high strength to weight ratio. Its superior chemical resistance, and high tensile and flexural strength are among the best FDM materials [17]. “Process-induced properties of FDM products” presents an effective approach for characterization of the evolved mechanical properties of ULTEM™ 9085 and verified by experimental observations.
Specimen Dimensions:

For this research three specimen were manufactured and tested. Specimens S1, S2 and S3 (Specimen1, Specimen2, Specimen3 respectively) share the same grid geometry. “Grid geometry” includes height and base of in-plane triangle formed by the ribs. As shown in Figure 8, the height of the triangle (h) was 1.130” and length of the side (a) was 1.310”. The specimen was initially modelled using SolidWorks and AutoCAD. The thickness of the ribs was 0.0259”.

---

**Figure 7 Mechanical and thermal properties of the ULTEM™ 9085[20]**

<table>
<thead>
<tr>
<th>Mechanical Properties¹</th>
<th>Test Method</th>
<th>English</th>
<th>Metric</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tensile Strength</td>
<td>ASTM D630</td>
<td>10,400 psi</td>
<td>71.6 MPa</td>
</tr>
<tr>
<td>Tensile Modulus</td>
<td>ASTM D630</td>
<td>32,200 psi</td>
<td>2200 MPa</td>
</tr>
<tr>
<td>Tensile Elongation</td>
<td>ASTM D630</td>
<td>6%</td>
<td>6%</td>
</tr>
<tr>
<td>Flexural Strength</td>
<td>ASTM D790</td>
<td>16,700 psi</td>
<td>115.1 MPa</td>
</tr>
<tr>
<td>Flexural Modulus</td>
<td>ASTM D790</td>
<td>342,600 psi</td>
<td>2500 MPa</td>
</tr>
<tr>
<td>IZOD Impact, notched</td>
<td>ASTM D256</td>
<td>2.0 ft-lb</td>
<td>106 J/m</td>
</tr>
<tr>
<td>IZOD Impact, un-notched</td>
<td>ASTM D256</td>
<td>11.5 ft-lb</td>
<td>613.8 J/m</td>
</tr>
<tr>
<td>Compression Strength</td>
<td>ASTM D695</td>
<td>15,200 psi</td>
<td>104 MPa</td>
</tr>
<tr>
<td>Compression Modulus</td>
<td>ASTM D732</td>
<td>280,000 psi</td>
<td>1930 MPa</td>
</tr>
<tr>
<td>Shear Strength</td>
<td>ASTM D732</td>
<td>8,300 psi</td>
<td>57 MPa</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Thermal Properties²</th>
<th>Test Method</th>
<th>English</th>
<th>Metric</th>
</tr>
</thead>
<tbody>
<tr>
<td>Heat Deflection (HDT)</td>
<td>ASTM D648</td>
<td>307°F</td>
<td>153°C</td>
</tr>
<tr>
<td>Heat Deflection (HDT)</td>
<td>ASTM D648</td>
<td>307°F</td>
<td>153°C</td>
</tr>
<tr>
<td>Glass Transition Temp</td>
<td>DSC (SYST)</td>
<td>367°F</td>
<td>186°C</td>
</tr>
<tr>
<td>Coefficient of Thermal Expansion</td>
<td>ASTM E228</td>
<td>3.57 x 10⁻⁶ in²/in°F</td>
<td>65.27 μm/(m·°C)</td>
</tr>
<tr>
<td>Melting Point</td>
<td></td>
<td>Not Applicable²</td>
<td>Not Applicable²</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Electrical Properties³</th>
<th>Test Method</th>
<th>Value Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>Volume Resistivity</td>
<td>ASTM D257</td>
<td>4.9 x 10⁻¹² ohms·cm</td>
</tr>
<tr>
<td>Dielectric Constant</td>
<td>ASTM D150-98</td>
<td>3.2 - 3.0</td>
</tr>
<tr>
<td>Dissipation Factor</td>
<td>ASTM D150-98</td>
<td>.0027 - .0026</td>
</tr>
<tr>
<td>Dielectric Strength</td>
<td>ASTM D149-99, Method A</td>
<td>250 - 110 V/mil</td>
</tr>
</tbody>
</table>
Specimen S1 (Figure 8) had rectangular dimensions of 4” x 4.6280” with a face sheet thickness of 0.12” and a rib height of 0.2”. Specimen S2 has similar rectangular dimension as S1 i.e. 4” x 4.6280” with a face sheet thickness of 0.15” and rib thickness of 0.25”. Specimen S3 (Figure 9) had a different dimensions, 4.75” x 7.25” with face sheet thickness 0.01 and rib height of 0.0189”.

Figure 8 Dimensions of Specimen S1
Figure 9 Dimensions of Specimen S3
DIC Testing Conditions:

“Digital Image Correlation” (DIC) “is an optical method that employs tracking and image registration techniques for accurate 2D and 3D measurements of changes in images”. This technique is used to measure deformation, strain, and optical flow. This technique has been increasing in popularity due to its relative ease of implementation and full-field data acquisition. Advancing technology and digital cameras are enabling technologies for this method. In this method images are recorded and processed using statistical correlation algorithm. Commercial software like Istra4D used in this work, provides turn-key operation of DIC systems. In order to obtain accurate results calibrating the DIC is an important step. Using the calibration data, the DIC system converts the image coordinates to geometric coordinates.

Using the geometric data, DIC is capable of providing true strains. All the data and images stored are utilized to obtain the required data. One of many advantages in DIC is, to calculate strains, deformation or any other data. The DIC doesn’t needs any kind of strain gauge setup. Results obtained will be accurate as long as the calibration is accurate. This thesis uses the DIC data along with the finite element analysis and analytical methods to better, predict and validate the buckling load of the Isogrid structure manufactured using the DFM technique. This procedures will be discussed in later sections. Figure 10 shows the experimental setup used during calibration and during the testing. One can easily observe the cameras used to record the image steps, the load frame and control and setup of specimen held between the two flat top/bottom plates. Figure 11 shows a specimen with speckle pattern and the DIC system with two cameras capturing the specimen images.
Figure 10 DIC experimental setup
Specimen were then spray painted with white and black dots on the surface facing the cameras. This technique provide some physical points that will be tracked by the cameras used in the test. Around the point of interest of the specimen a square subset of pixels are identified on the speckle pattern on the reference image and their corresponding location are determined on the deformed image.

Figure 11 Specimen S1 and S2

Figure 12 Specimen S3
**Loading and Boundary conditions.**

Figures 13, and 14 show the test setup for compression (buckling) testing. Specimens S1 and S2 were tested under identical conditions. The load was applied along the length of the sample, such that load was primarily carried by the skin and ribs along their direction. Sample S3 was tested in a similar way but along different orientation.

Load was applied on the base of the triangle formed by the ribs. Throughout the literature it was stated that the isogrid provide a greater resistance when load is applied along its rib rather than its base. The literature also stated that as the ratio of thickness of ribs to thickness of skin increases buckling of isogrid will shift from ribs to skin. Hence, this research has considered two specimen with two different skin thickness S1 and S2 and one with extremely thin skin but tested along a different directions.

Boundary conditions were the same for all samples. As shown in Figures 13 and 14 the two opposite ends was restrained with thin metallic strip to prevent slip on the end plate. The face simply supported was designated as “a” and opposite face taking as “b”. Side “a” is clipped which will restrict the motion of this face in x and z directions (taking the loading direction as “x”). Boundary conditions were defined at (x, z) = (0, 0).

Pressure acts in the y direction on face. The compressing piston was set at a rate of 0.05 inches per minute. These boundary conditions were same for all the specimen. Machine setup, boundary conditions and loads can be seen in the Figures 13 and 14.

All specimen were loaded along the longest side of the sample. Short end faces were free. S1 and S2 were compressed between plates along its longest edge i.e. the 4.75 in” edge and S3 was compressed along its longest edge 7.25 in”.
Figure 14 Loading directions and boundary conditions on S1 and S2
Finite Element Analysis Procedure:

Finite Element Analysis (FEA) is a numerical method to predict the response of material and structures to real world forces like stress, pressure, vibration, heat flow and other physical effects. This method can numerically predict if the specimen will break, bend, buckle, wear out or work at given conditions. This technique can be used to estimate the efficiency of the part before it’s actually used and sometimes to observe and explain real observations. This technique approximates solutions to boundary value problems to partial differential equation. The physical model is divided into simple mathematical elements called as finite elements [18].

In order to obtain the most accurate results the numerical model should have the same environment of the real world models. “Environment” in this context includes the boundary conditions, loads, temperature, and body forces. The closer these conditions match real conditions, the more accurate will be the results. Most of the time the model tested in the real world may not be as perfect as the numerical model. Hence, the real world will involve some “imperfections”. All these procedures are adopted to obtain a solution that is more accurate.

FEA analysis dates back to 1960 and 1970s. Since then there are n number of software packages available to perform this analysis to include a few COMSOL, Nastran Patran, Strand7™, ABAQUS, Autodesk and many more. This research uses the Strand7™ [24] as the analysis software. This is a simple FEA analysis package developed in 1988. It is now being vastly used in several applications including mechanical, civil, structural, aeronautical and other engineering fields. Developers define Strand7™ as “fully-integrated visual environment - combined with a suite of powerful
solvers - gives you unparalleled functionality in a single application. Construct models, run analyses and investigate results simultaneously using a seamless interface”.

In order to overcome any possible modeling mistakes, the 2D model from SolidWorks is exported into Strand7 and extruded to form the 3D shape. Skin and ribs were extruded to their thickness simultaneously. Material properties were assigned to the part. The part was finely meshed using Quad8 elements. Element continuity and displacement compatibility was maintained. Once the part is meshed load and boundary conditions were applied. As discussed in the section the Load and Boundary conditions were applied on the top and bottom either faces and restricted along ‘y’ and ‘z’
Both top and bottom faces of the part are restricted along ‘y’ and ‘z’ i.e. ‘0’. And surface pressure of 10 psi is applied on the top face, i.e. x = -10. In order to avoid rigid body motion, couple of nodes along the ribs are restricted in ‘x=0’; Figure 16 indicates all the boundary conditions on the face. The other picture here shows both the boundary conditions and loading conditions.
Buckling values are calculated from linear static solver followed by linear buckling solver. Over ten Eigen values were obtained. First Eigen value times the load applied gives us the value at first buckling mode. Total number of 18,799 elements were created. Same loading conditions and boundary conditions were applied to the other two specimen. Figure 16 shows the Isogrid with loads and boundary conditions. Figure 17 shows the specimen S3 with loads and boundary conditions.
Figure 17 Loading and boundary conditions for Specimen S1 and S2
Figure 18 Loading and boundary conditions for Specimen S3
Closed-Form Approach:

The literature review section of this work discussed several different approaches for buckling stresses, strains and other mechanical properties of different grid structures. Here this report has adopted the technique developed by Stephen W Tsai, PI in their paper “Manufacture and Design of Composite Grids”. Stiffness of Quasi-Isotropic Laminates and equivalent grids are compared. With equally spaced ply orientation of \([\pi/3], [\pi/4], [\pi/5]\) laminates becomes quasi-isotropic. This is applied to isotropy of grids when three are oriented at an angle of 60\(^\circ\) to each other. Invariants of quasi-isotropic are linear combinations of the ply stiffness components are shown below in the equation set [1]

\[
U_1 = \frac{3}{8} (Q_{xx} + Q_{yy}) + \frac{1}{4} Q_{xy} + \frac{1}{2} Q_{ss}
\]

\[
U_4 = \frac{1}{8} (Q_{xx} + Q_{yy}) - \frac{1}{4} Q_{xy} + \frac{1}{2} Q_{ss}
\]

\[
U_5 = \frac{1}{8} (Q_{xx} + Q_{yy}) - \frac{1}{4} Q_{xy} + \frac{1}{2} Q_{ss}
\]

Young’s modulus, Poisson’s ratio and Shear modulus of laminates of quasi-isotropic laminates in terms of invariants are

\[
E^{[iso]} = \frac{D}{U_1}, \nu^{[iso]} = U_4/ U_1, G^{[iso]} = U_5; \text{ where } D = U_1^2 - U_4^2
\]  

[2]

When the degree of anisotropy of a composite ply increases to upper limit longitudinal stiffness \(E_x\) would be the dominant component. Matrix related components become negligible and invariants and resulting engineering constants from this approach are

\[
U_1 = \frac{3}{8} E_x, \quad U_4 = \frac{1}{8} E_x, \quad U_5 = \frac{1}{8} E_x
\]

[3]
\[ v^{[iso]} = \frac{1}{3}, G^{[iso]} = \frac{1}{8} E_x, D = \frac{1}{8} E_x^2, E^{[iso]} = \frac{1}{3} E_x \] [4]

\( E_x \) in the last equation may be explained physically by viewing at a laminate having three independent plies of equal thickness. Each ply thickness would be 1/3\(^{rd} \) of total laminate thickness, so effective stiffness is equal to 1/3\(^{rd} \) of unidirectional stiffness. Same stiffness plies at 60\(^{o} \) intervals makes the laminate isotropic.

According to this report a composite isogrid as a regrouped laminate shown in the image below where the matrix stiffness is approaching zero. 1/3 factor as in equation [4] can be applied for the contribution of rib stiffness to the isogrid stiffness. Global stiffness however depends on rib area fraction ‘f’. This fraction is same as rib volume fraction if grid pattern remains constant along the grid height. Value of \( f_{iso} = \frac{2\sqrt{3}}{L} \), here factor is directly proportional to the ratio of the width ‘b’ and spacing ‘L’ of each rib.

Figure 19 Grid terminology

Poisson’s ratio of isogrid is 1/3 same as a laminate without matrix. Global young’s modulus and poisson’s ratio are given below which are easy to use [5]. These relations are applied to interlaced isogrids where all ribs are in the same plane. Global thickness of the grid would be the sum of the rib thicknesses. Stacked joint ribs run in different orientations run in different planes and effective stiffness will be lower than that.
of the interlaced grid given same overall rib geometry. So, interlaced grid have good
effective stiffness when compared to stacked joint ribs. Grid stiffness depends on rib
stiffness and geometry. Rib intersection either pinned or fixed does not affect the grid
stiffness if the slenderness ration is high. This is the case when the area fraction is small.
If ply composite properties are moderately anisotropic matrix related components are no
longer negligible. Quasi-isotropic laminate stiffness in equation [2] are applicable.

\[ E^{[\text{isogrid}]} = \frac{L}{3} E_x = \frac{2 (L/b)}{\sqrt{3}} E_x \; ; \; v^{[\text{isogrid}]} = 1/3 \]  

[5]

Grids can be designed with either one or two skins. Grids are capable of carrying
the applied external in-plane and flexural loads, while skin share the load. Skin and ribs
interaction makes this analysis a little complicated. Buckling of the ribs are constrained
by the skin and additional skin buckling failure mode is added to this. Rule of mixtures
can be used if skin and ribs are made of two different material.

This research has utilized both laminate and grid properties in order to calculate
the effective buckling stiffness or the isogrid manufactured. Skin is treated as laminates
and equations [1] and [2] were utilized to find the effective properties of skin. Equation
[5] has been utilized to calculate the effective properties of the grid structure. Effective
thickness will be thickness of the ribs. Here we now have varying properties of the skin
and ribs which act like two different material. Rule of mixtures will be applied to
calculate the effective young’s modulus and poisson’s ration of the plate. Buckling
equation of the plate is given by

\[ P_x = b \frac{a^2 n^2 D}{m^2} \left( \frac{m^2}{a^2} + \frac{n^2}{b^2} \right)^2 \]  

[6]

\[ D = \frac{E t^3}{12(1-v^2)} \]  

[7]
CHAPTER 4
RESULTS AND DISCUSSION

Results in this Chapter are discussed on a sample-by-sample basis. For each sample, experimental DIC results are discussed, followed by finite element results, and finally results of applying closed-form solutions from the literature. A summary of the results will follow individual specimen discussion

Specimen S1

DIC results:

Specimen S1, S2 and S3 were compressed uniaxial. ISTRA 4D [26] is a user friendly software package designed for digital image correlation, analysis and evaluation. Correlation (Calibration) series, which were recorded during each experiment were be imported into the software for comparison during the buckling experiments. Specimen S1 had a total of 228 correlation series steps, S2 had 208 steps and S3 has 58 steps. Each of these steps has data from measuring the changes of approximately ten thousand grid point and more. These grid points are correlated with the reference step (which may be the first step where no forces are acting) to calculate the desired value of displacement or strain.
Each step has load, displacement and strain stored in x, y and z directions. Average, minimum, or maximum values can also be computed and stored. To observe the first buckling mode, a time and load graph has to be considered. Since buckling is state of instability, sudden changes in load (for a constant displacement test), displacement, or strain are indications of a buckling event. Figure 20, 21 graphs that are plotted using Matlab, a general purpose computation, symbolic algebra and plotting package.
Figure 20 Step vs Load graph Specimen S1

Figure 21 Step vs displacement in 'z' graph Specimen S1
As can be seen in Figure 20, specimen loading was stopped soon after a rapid z-displacement increases, indicated the onset of buckling. Hence maximum load is taken as the buckling load and it is 797.462 lbs. Time vs load gives us the value of load at that time. Simultaneously x, y and z displacements of the specimen are recorded at respective each. This “time” at certain “step” will give us the buckling shape, displacement of the specimen in x, y and z direction and respective strain values. Since here the skin thickness is dominant and specimen S1 is not allowed to buckle completely the modes might not be seen clearly.

Figure 21 shows the displacement field of Specimen S1 at predicted buckling load and buckling values obtained here are to be compared with Finite Element analysis and Analytical solutions later in this section.

*Figure 22 Displacement of Specimen S1 in 'z'*
Finite Element Analysis Results:

Modelling, analysis, boundary conditions and loading conditions were discussed in Chapter 3 under Finite Element Analysis procedure section. Specimen S1 has 18788 elements. Solver in Strand7™ set to calculate first 10 eigen values and modes at 10lbs load. Since 10 lbs. of load was applied to the model buckling loads were 10 times the eigen values compared by the solver. Table 1 gives all the ten buckling values.

Table 1 FEA buckling values for specimen S1

<table>
<thead>
<tr>
<th>Mode Number</th>
<th>Mode value in lbs.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>83.1192322</td>
</tr>
<tr>
<td>2</td>
<td>195.318124</td>
</tr>
<tr>
<td>3</td>
<td>273.196002</td>
</tr>
<tr>
<td>4</td>
<td>413.359416</td>
</tr>
<tr>
<td>5</td>
<td>526.741010</td>
</tr>
<tr>
<td>6</td>
<td>562.557565</td>
</tr>
<tr>
<td>7</td>
<td>582.654594</td>
</tr>
<tr>
<td>8</td>
<td>647.196899</td>
</tr>
<tr>
<td>9</td>
<td>743.703111</td>
</tr>
<tr>
<td>10</td>
<td>755.263021</td>
</tr>
</tbody>
</table>

Table indicates first ten buckling values. First buckling mode will be at 831.19 lbs. will be considered as buckling load. Figure 23 indicates the first ten buckling mode of specimen S1. It can be observed that first six buckling modes are global buckling
mode, and no rib crippling observed till 7th buckling mode. Because of thicker skin rib buckling failure mode of the structure could be predicted because of rib failure.
Figure 23 1-4 buckling modes of Specimen S1
Closed-Form analysis results:

Equations 1-5 were used to calculate the buckling load for closed form results.

Material properties used in these equations as obtained from [12] are shown in Table 2

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_1$</td>
<td>3.68 e05 psi</td>
</tr>
<tr>
<td>$E_2$</td>
<td>3.37 e05 psi</td>
</tr>
<tr>
<td>$E_3$</td>
<td>3.64 e05 psi</td>
</tr>
<tr>
<td>$G_{12}$</td>
<td>9.21 e04 psi</td>
</tr>
<tr>
<td>$G_{13}$</td>
<td>9.217 e04 psi</td>
</tr>
<tr>
<td>$G_{23}$</td>
<td>8.49 e04 psi</td>
</tr>
<tr>
<td>$V_{12}$</td>
<td>0.46</td>
</tr>
<tr>
<td>$V_{13}$</td>
<td>0.39</td>
</tr>
<tr>
<td>$V_{23}$</td>
<td>0.40</td>
</tr>
</tbody>
</table>

Table 2 Material properties for all Specimen S1, S2 and S3

The above property values were used for all specimen. The Matlab code used to calculate closed-form buckling loads is listed below.
clear all
clc
% all values are in PSI system %
L = 1.28304;
b = 0.0259;
% stiffness material properties
E1 = 368308.73538;
E2 = 337633.216183;
E3 = 365436.9889;
v12 = 0.46;
v13 = 0.39;
v23 = 0.40;
G12 = 92171.45835;
G13 = 92171.45835;
G23 = 84965.85414;

F = 1-((v12)^2*(E2/E1)); %denominatior of Q11&Q22
Q11 = E1/F;
Q22 = E2/F;
Q12 = v12*Q22;
Q66 = G12;
%strain transformation matrix
m = cos(45);
n = sin(45);
% angle transformation matrix will be denominated by A
A = [m^2 n^2 -2*m*n;
    n^2 m^2 2*m*n;
    m*n -m*n (m^2 - n^2)];
E = [Q11 Q12 0;
     Q12 Q22 0;
     0 0 Q66];
Q = A * E * transpose(A);
\begin{verbatim}
Q1 = Q(1,1)+Q(2,2);
Q2 = Q(1,2);
Q3 = Q(3,3);
% Stephen Tsai's paper method for Equivalent thickness and stiffness
U1 = (Q1*(3/8))+(Q2 /4)+(Q3 /2);
U4 = (Q1 /8)+(Q2*(3/4))-(Q3 /2);
U5 = Q1 /8-(Q2 /4)+Q3 /2;

D = U1^2 - U4^2;
%laminate properties
E_1 = D/U1
ν_1 = U4/U1
G_1 = U5

%rib_area fraction

f = (2/sqrt(3))*(L/b);

E_2 = f* E1
ν_2 = (1/3);

%rule of mixtures
%E_iso = (E_1+E_2)
%ν_iso = (ν_1+ν_2)
\end{verbatim}
This code provides the Young’s modulus, Poisson’s ration and values for ribs and skin. As per “Manufacture and Design of Composite grids” by Stephen W. Tsai [15] effective properties of grid structure along with skin are calculated using rule of mixtures. This present work utilized Autodesk Simulation Composite Design 2014 package to find the effective material properties of all specimens. $E_1$, $\nu_1$ and $G_1$ are skin properties i.e. laminate properties $E_2$ and $\nu_2$ are rib properties. Boundary conditions used while testing were neither completely simply supported nor clamped. Simply supported is considered the closest boundary condition. According to given value of critical load is 896.88 lbs.

**Specimen S2**

**DIC results:**

Specimen S2 has all the same dimensions apart from skin and rib thickness. Skin thickness was increased from 0.12 in to 0.15 in whereas ribs were increased from 0.2 in to 0.25in. Skin to rib height ratio is maintained at the same value as specimen S1. Specimen S2 has similar boundary and loading conditions as S1. Specimen S2 was tested with 209 steps. The step vs load is given in the figure 23 below. As seen in this figure until and around step 100 load was constantly increasing. As shown in Figure 24 at step 89 the slope of the z-displacement changes significantly from its original value, which is taken as an indication of buckling. As shown in Figure 23, load at this step was 1620 lb. which is taken as the critical buckling load.
Figure 24 Step vs Load graph Specimen S2

Figure 25 Step vs Displacement in ‘z’ graph Specimen S2
Figure 26 shows indicates the displacement of specimen at critical buckling model. This can be useful to compare the first buckling mode for finite element analysis. Though displacements are negligible it can still be observed that specimen is slightly bend in at the center. This will be considered as its first buckling mode.

*Figure 26 Displacement of Specimen S2 in ‘z’*
Finite Element Analysis results:

Specimen S2 has the same boundary conditions and loading as specimen S1. He FEA model of specimen has a total of 21504 elements and 21457 nodes. The element types for ribs and skin were the same in the S2 model as in the S1 model. Since 10 lbs. of load was applied all eigen values must be multiplied by 10 to obtain the respective buckling loads for each eigen value. The solver was again set to calculate 10 eigen values along with eigen modes. All ten eigen values for specimen S2 are listed in the table below. Table 3 it can be understood that first buckling mode occurs at 169 lbs. according to finite element analysis. All ten buckling modes of specimen 2 are shown in Figure 25 below.

<table>
<thead>
<tr>
<th>Mode Number</th>
<th>Mode value.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>169</td>
</tr>
<tr>
<td>2</td>
<td>444</td>
</tr>
<tr>
<td>3</td>
<td>625</td>
</tr>
<tr>
<td>4</td>
<td>632</td>
</tr>
<tr>
<td>5</td>
<td>754</td>
</tr>
<tr>
<td>6</td>
<td>833</td>
</tr>
<tr>
<td>7</td>
<td>833</td>
</tr>
<tr>
<td>8</td>
<td>843</td>
</tr>
<tr>
<td>9</td>
<td>845</td>
</tr>
<tr>
<td>10</td>
<td>852</td>
</tr>
</tbody>
</table>
Figure 27 1-4 buckling modes of specimen S2
Figure 27 indicates all ten eigen modes. First eigen mode when compared to figure 26, it can observed that the first critical buckling modes of both DIC and FEA analysis were the same. Since thickness of S2 is much more S1 rib crippling is prominent and occurred early when compared to specimen S1.

**Closed-Form analysis results:**

Equation 1 -5 lead to effective Young’s modulus and poisson’s ratio of rib and skin individually. Matlab code and material properties from table 4 for ULTEM can be used to calculate the loads. Simulation Composite Design software was used to calculate loads for specimen 2. The closed-form solution gives a buckling load of 1751 lbs.

**Specimen S3**

**DIC results:**

Specimen S3 orientation during loading and for boundary conditions was different from specimen S1 and S2. Loads will be applied along the base of triangles. Specimen S3 was painted on the rib side in order to study rib and skin interaction. This specimen was unable to carry heavy loads. Total data was recorded in 80 number of steps. 278 lbs. was the maximum load applied to this specimen. Figure 26 gives us the graph for displacement in ‘z’ direction and step. Between step 23 and 30 it can be observed that displacement fluctuates rapidly with respect to load. Load around step 23 is 41.096 lbs.
Figure 28 Step vs Load Specimen S3

Figure 29 Step vs Displacement in ‘z’ direction Specimen S3
Figure 30 indicates the displacement of specimen at its critical load. As we can in this case the first buckling mode was skin buckling. Since skin in specimen S3 is thin it could be expected that pocket buckling mode will the critical buckling mode.

Figure 30 Displacement of Specimen S3 in ‘z’
Finite Element Analysis results:

Modelling, loading and boundary conditions were discussed in Chapter 3. Specimen S3 has a total of 29405 nodes and 29520 elements. Grid geometry is maintained constant for all three specimen. In order to observe the behavior of skin specimen S3 was made of thin skin. Table 10 provides us the eigen values and respective eigen modes. Figure 28 indicates ten different modes of buckling. Since 10lbs of load is acting, buckling load will be 10 times the first Eigen value. Table 4 gives us the buckling load for specimen S3.

<table>
<thead>
<tr>
<th>Mode Number</th>
<th>Mode value.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4.1774</td>
</tr>
<tr>
<td>2</td>
<td>4.9008</td>
</tr>
<tr>
<td>3</td>
<td>5.6144</td>
</tr>
<tr>
<td>4</td>
<td>6.0926</td>
</tr>
<tr>
<td>5</td>
<td>6.1149</td>
</tr>
<tr>
<td>6</td>
<td>6.2778</td>
</tr>
<tr>
<td>7</td>
<td>6.7128</td>
</tr>
<tr>
<td>8</td>
<td>7.8218</td>
</tr>
<tr>
<td>9</td>
<td>8.1650</td>
</tr>
<tr>
<td>10</td>
<td>8.2796</td>
</tr>
</tbody>
</table>
Figure 31.1 – 4 buckling modes for specimen S3
Closed-Form Analysis results:

In chapter 3 analytical analysis has provided relationship for material and structural properties. Since the loads are acting against the base, it is only distributed along its ribs and hence specimen cannot support heavy loads. This has been noted. Specimen S3 proves this point. Analytical results gives us a value of 538 lbs. for the buckling load.

Discussion

Specimen S1, S2 and S3 were additively manufactured using fused deposition modelling technique. Specimen ribs were at 60º angle to each other to form an equilateral triangle. Displacement and strain were measured and analyzed using digital image correlation techniques and the Istra4D software package. These values were then compared with Finite Element Analysis (Strand7 software package) and a closed-form procedure adapted from “Manufacture and Design of Composite Grids” to calculate grid properties. Buckling values of three specimen calculated in three methods are summarized in table 5.

<table>
<thead>
<tr>
<th>Specimen</th>
<th>DIC results</th>
<th>FEA results</th>
<th>Analytical results</th>
</tr>
</thead>
<tbody>
<tr>
<td>S1</td>
<td>797.462 lbs.</td>
<td>831.97 lbs.</td>
<td>896.88 lbs.</td>
</tr>
<tr>
<td>S2</td>
<td>1621.49 lbs.</td>
<td>1696.30 lbs.</td>
<td>1751.94 lbs.</td>
</tr>
<tr>
<td>S3</td>
<td>41.096 lbs.</td>
<td>41.774 lbs.</td>
<td>538.4 lbs.</td>
</tr>
</tbody>
</table>
All specimens' first buckling mode can be observed both in its DIC and FEA results. Figure 32, 33 and 34 represent the first buckling modes from DIC and FEA for Specimen S1, S2 and S3 respectively. It can be observed that for both specimens S1 and S2 displacement is in negative z direction.

It can also be observed that as the thickness increases the buckling value of the specimen will increase and buckling modes transfer from skin to ribs. Specimen S2 which has high thickness compared to Specimen 3, fails due to rib crippling. Figure 35 indicates the failed ribs for specimen S2. Specimen S3 indicates the local buckling modes in skin before failure due its thin skin values. Specimen S3 also demonstrates that when specimen is loaded normal to the longitudinal ribs transmission of load is not as efficient as when it is loaded along the ribs. These images demonstrate that obtained analytical results were valid and verified.
Figure 32 First buckling modes from DIC and FEA Specimen S1

Figure 33 First buckling modes from DIC and FEA Specimen S2
Figure 34 First buckling modes from DIC and FEA Specimen S3

Figure 35 Rib Failure Specimen S2
Table 6  Percentage error of FEA and Closed form approach to DIC results

<table>
<thead>
<tr>
<th>Specimen</th>
<th>%error of FEA</th>
<th>%error of Analytical procedure</th>
</tr>
</thead>
<tbody>
<tr>
<td>S1</td>
<td>4.14</td>
<td>11.08</td>
</tr>
<tr>
<td>S2</td>
<td>4.61</td>
<td>8.04</td>
</tr>
</tbody>
</table>

Table 14 provides the percentage error of buckling loads for FEA and closed form approach analysis with respect to the experimental value. It helps to realize the efficiency and accuracy of the approach. The reason for %error of S1 and S2 for closed-form analysis can be explained through applied boundary conditions and assumed boundary conditions. For all the specimens, the boundary conditions in the experiment were not specifically SSSS (four sides simply supported) or SSCC (Two sides simply supported and two sides are clamped), as used in the analysis. The closed-form analysis assumed that the plate was simply supported all the four sides which was considered to be the most similar to the boundary conditions in the experiment. The percentage error of S3 for closed-form analysis is extremely large, as the approach has not considered the application of loads against the ribs.
The objective of this research was to build and experimentally characterize a 3D-printed isogrid structure for uniaxial buckling behavior, and compare the loads and modes to those obtained analytically using closed-form and finite element analyses...

Isogrid specimens built using Fused Deposition Modeling (FDM) were subjected to uniaxial compression loading with approximated simple supports. Strain and displacement data were captured using Digital Image Correlation (DIC), Observed loads and buckling mode results were successfully compared to Finite Element Analyses (FEA) and closed-form analytical procedure adopted from analyses of composite structure.

Uniaxial compression was conducted with approximated simple-support boundary conditions. These loading and boundary conditions were modeled as purely simple supports during FEA analysis. Hence, an error of 1% - 4% was introduced into the comparison of experimental and FEA results. It was also demonstrated that when load was applied along the longitudinal stiffeners, the structure requires more load to buckle than compared to loading transverse to these stiffeners.
Since load applied transversely to the longitudinal stiffeners, Specimen S3 could not hold a substantial load. This observation was validated in both DIC and FEA results (the closed form solution did not consider the case of loading transversely to the longitudinal stiffeners).

This is proved in both DIC and FEA results. Analytical analysis has not discuss the case of varying load directions. Results vary by a scale of five hundred. The load orientation into the isogrid model did not change the results from the analytical method. Percentage error in analytical analysis were so prominent because of the boundary conditions. Research has considered simply supported boundary conditions to a simpler boundary conditions.

Finite Element analysis considered the model as ideal as a geometric model with no imperfections. But this has to be noted that while manufacturing the model, in order to obtain the closest possible rib with raster width of 0.02” was chosen. While ribs were of 0.03” width. Hence there are many possible chances where the physical specimen is not as perfect as the numerical model used in analysis. If imperfections were introduced possibilities of obtaining more accurate results would have been there. Linear buckling analysis was used to obtain the critical buckling values, where non-linear analysis could have given better results.

With all tabled results it can be concluded that analytical analysis and properties of composites can still be used for an additively manufactured isogrid. Proper boundary and loading conditions will results in more accurate results.
**Recommended Future Work:**

Varying closed form solutions along with varying loading conditions should be pursued to expand the realm of validated existing solutions that can be applied to FDM structure. Larger isogrid structure with thicker stiffeners may require different toolpaths, resulting in changes to material properties that should be investigated (the degree of anisotropy of the current samples is relatively low. The effect of filleted intersections between the stiffeners and skin should be investigated as a way to extend the buckling performance with little additional mass, since such fillets can be easily introduced in a 3D-printed structure. The role of imperfections in 3D printed structure (void content is difficult to keep below the range of 7-10%) in determining strength of post-buckled structure, particularly within the intersection of stiffeners, is also worth investigating.
BIBLIOGRAPHY

1. Website (www.3ders.org, n.d.) 3D printer and 3D printing news


19. SolidWorks, Dassault Systèmes SOLIDWORKS Corp, December 1993
26. Istra4D Software, Dantec Dynamics A/S.