Methods to Streamline Laminate Composite Analysis and Optimization

Ammon I. Hepworth[1], C. Greg Jensen[2] and James T. Roach[3]

[1]Brigham Young University, ammon.hepworth@gmail.com
[2]Brigham Young University, cjensen@byu.edu
[3]Pratt & Whitney, james.roach@pw.utc.com

ABSTRACT

Major market growth in the area of advanced composite materials has spurred the development of methods to streamline the analysis and optimization of these materials. A computer tool has been developed to automatically create a finite element model with greater detail than previous zone-based methods. This tool can save practitioners dozens of hours during the pre-processing stage of analysis. It can also be used in an optimization to find the best possible lay-up for a laminate composite part. A method to automatically apply boundary conditions and solve the finite element model is also presented. This method is used in conjunction with the automated composite finite element tool in an iSight optimization to optimize the modal frequencies of a composite wing model.

Keywords: laminate composite analysis, automation, optimization.
DOI: 10.3722/cadaps.2010.377-386

1 INTRODUCTION

In recent years there has been major market growth in the area of advanced composite materials [13]. The aerospace industry is especially taking advantage of the high strength and low weight characteristics of these materials. Despite the fact that composites are becoming so widespread, advanced composite material design, analysis and manufacturing is still a relatively new field and is generally still a time consuming process. In order to speed-up the design, analysis and manufacturing of composite materials, methods have been developed to streamline these processes. In a previous paper, the authors discussed methods to automate the design of composite materials [5]. This paper will discuss methods to streamline composite analysis and optimization.

Traditionally, high performance composite materials are generally fabricated using high modulus fibers imbedded in a lower modulus resin. Often, these composite materials are formed by stacking several fiber dense layers to build up the 3-dimensional part. Composite parts that are made using this stacking procedure are called laminated composites, due to the fact that they are made of several layers, or lamina. Each layer of the stack is positioned in such a way to allow designers to customize the material properties of the part to meet the needs of a specific structural application. For instance, if a structural part needs higher strength in one direction in order to be able to support a given load,

more lamina are positioned such that their fiber orientation angles are pointing in the direction that best resists that load [2, 11].

Due to the plethora of combinations of lamina orientation angles in a multi-layer laminate, optimization is a powerful tool to explore the design space and find the best possible lay-up for a composite part [3]. To make optimization possible and yield accurate results, a fast and reliable analysis method is essential. For composite parts with complex geometries, finite element modeling is the preferred analysis method [2]. However, complex finite element models having multiple layers with unique orientation angles are time consuming to set up. The pre-processing phase is also quite tedious, leading to many opportunities for error. In the past decade there have been some who have developed methods to streamline composites analysis and use them in optimization.

Vasey-Glandon et al. patented a computer program called the Parametric Composite Knowledge System (PACKS). This program generates an optimized 3-D ply definition for a laminate composite part. The program divides the plies into zones and uses FEA analysis software as well as predetermined design rules to optimize these zones to perform better in manufacturing [12]. PACKS reduced the cost of many aircraft composite development programs by over 60% and was licensed to Unigraphics Solutions, Inc for commercial use in 2001 [4].

Another composites optimization computer program, developed by Menayo et al., optimizes a ply lay-up for a composite part to form a preliminary ply model. This is done by first running a stress analysis program (called ARPA) to obtain a stacking sequence for divided regions of the part. It then automatically generates a ply stacking table that organizes the plies into zones. The stacking table is then optimized using design stacking rules and manufacturing constraints. Next, a ply drop-off distribution is performed to determine which plies end-up in a given zone and order them according to design rules [6].

Siemen's NX has an integrated, zone-based laminate composites application that allows for ply modeling, draping, and optimization within the CAD system. In ply modeling, it allows the user to associate ply lay-ups with NX surfaces (zones). Draping predicts ply orientations along doubly-curved surfaces making the analysis more accurate. The optimization piece allows users to optimize laminate properties to meet specific design criteria [10].

All of the methods discussed above use a similar approach to set up the laminate composite analytical model. They use a zone based method that divides the part into zones where each zone has a specific lay-up. The downside of this type of approach is that the analysis only looks at zones that approximate a stacking sequence for an entire region. In parts having geometry with rapidly changing curvature, this approximation may not suffice. The approach that is discussed in this paper uses the composite part geometry and a ply table to automatically create a finite element model that has a unique ply lay-up for each element in the finite element mesh. Thus, the model is created without having to divide the part into zones, giving more detail to the model. In addition to this automatic analysis method, an approach to composites optimization that utilizes this method is also discussed herein.

2   METHOD

The method discussed in this section will go through the process to optimize the lay-up of a component made from a multi-layered heterogeneous material. Each layer of the material may have unique material properties and thickness. All the layers are stored in a database with each one given a priority number based on how frequently that layer is used throughout the part. The component itself is stored as two non uniform rational B-Spline (NURBS) surfaces representing either side of the said component. It is assumed that the reader is familiar with the basics of NURBS mathematics, but for those desiring to know more information see [8] and [9]. This section will discuss critical aspects of the optimization method including:

1.   Generation of the finite element model

2. Assigning parametric boundary conditions
3. Optimization algorithm and fitness to drive to an optimum

2.1   Generation of the Finite Element Model

Two NURBS surfaces are used to define the outer surfaces of the component made from a multi-layered heterogeneous material. These surfaces are called: $\{S_1(u,v)\}$ and $\{S_2(u,v)\}$. The points that make-up the mid-surface $\{M_{i,j}\}$ of the two outer surfaces is found by discretizing each surface by incrementing through the $\{u,v\}$ parameters of the surface and then averaging the points on each surface as shown:

$$M_{i,j} = \sum_{i=0}^{m}\sum_{j=0}^{n} \frac{S_1(m,n) + S_2(m,n)}{2}$$

A mid-surface is created using an interpolating function to find the control points and knot vector for the associated NURBS surface. After that, a grid of points $\{G_{i,j}\}$ that is $m \times n$ is found on the mid-surface by incrementing through the $\{u,v\}$ parameters of the mid-surface as shown:

$$G_{i,j} = \sum_{i=0}^{m}\sum_{j=0}^{n} M(m,n)$$

This grid of points is connected in the following way to form quadrilateral shell elements $\{E_k\}$:

$$E_k = \sum_{k=0}^{(a-1)(b-1)} \begin{bmatrix} e_{1k} \\ e_{2k} \\ e_{3k} \\ e_{4k} \end{bmatrix} = \sum_{i=0}^{a-1}\sum_{j=0}^{b-1} \begin{bmatrix} G_{i,j} \\ G_{i+1,j} \\ G_{i+1,j+1} \\ G_{i,j+1} \end{bmatrix}$$

Next, the center point of each element $\{C_{i,j}\}$ is found by querying the points on the mid-surface halfway between the increments used to find $\{G_{i,j}\}$:

$$C_{i,j} = \sum_{i=0}^{m}\sum_{j=0}^{n} M(m/2, n/2)$$

The unit normal vector at the center of each element is found by solving for the normal vector of the mid-surface at each center point. The negative normal vectors are also found. This is done using the traditional formula [2]. Figure 1 shows a side view of a component with the top and bottom curves representing the opposing surfaces. The mid-surface is the curve shown in between them with positive and negative normal vectors taken at constant intervals.
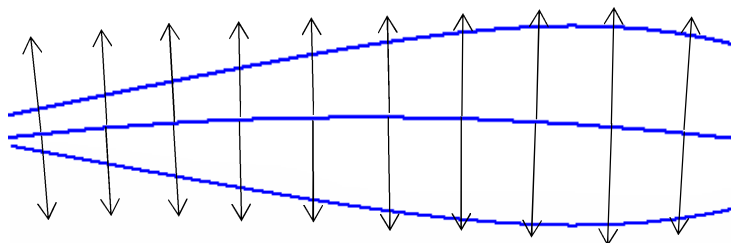

Fig. 1: A side view of the opposing surfaces, mid-surface and normal vectors.

The intersection points of one outer surface and each normal vector are then determined using intersection algorithms. The intersection points of the other outer surface and each negative normal vector are also found. The distance between the two intersection points on either surface is approximated as the thickness of the part at a given element. To obtain each element laminate definition, the layers are pulled from the database one by one, by priority, and the thicknesses of the layers are added up until it reaches the element thickness. All the elements are iterated through to find the layers that are assigned them. Once this is done the heterogeneous finite element model is complete.

2.2   Parametric Boundary Conditions

In addition to the automatic generation of the finite element model, another key component of the optimization is a method to automatically assign boundary conditions to the finite element model including loads and constraints. This is done using an algorithm that assigns loads or constraints to nodes based on their desired location on the part. For instance if the desired location of a boundary condition is along the bottom end of the part, boundary conditions $\{B_i\}$ are assigned to mid-surface nodes $\{G_{i,j}\}$ of size $m \times n$ in the following way:

$$B_i = \sum_{i=0}^{m} G_{i,0}$$

If the desired location of a boundary condition is along the left side of the part, boundary conditions of the same part are assigned as follows:

$$B_i = \sum_{i=0}^{n} G_{0,m*j}$$

These two examples demonstrate the feasibility of an automatic assignment of boundary conditions. It can also be shown that many other similar algorithms may be performed based on a different desired location of these boundary conditions with respect to the nodes; however they will not be explored in this paper.

2.3   Optimization Algorithm and Fitness of Design

The purpose of the optimization algorithm is to adjust the lamina properties in the database to determine the design that best fits a specific load case. The chosen algorithm to do this is a genetic algorithm which is based on Darwin's theory of evolution [7]. The primary reason why a genetic algorithm was chosen over a gradient based method is because gradient based methods were frequently unable to find an optimum with the large numbers of design variables required in this optimization. With a genetic algorithm, all of the design variables are filled with random values and the design evolutionarily moves towards a global optimum. This allows for the use of many design variables because all the variables iterate simultaneously instead of one by one.

One critical aspect that is needed when using a genetic algorithm is having some sort of criteria that determines how well a specific design performs as compared to other designs. This is called the goodness of fit or fitness factor. For example, in the case of an optimization where the objective is to reduce the stress in a part, the fitness factor can be as simple as the resulting stress output from the finite element analysis. In different, not so simple analyses where there are several outputs from the FEA, a fitness factor can to be determined from an average of all the results. In other cases it may be more effective to take the minimum or maximum of these results. Note that it most likely will take some experimentation to determine the best method to find fitness factor in more complicated analyses.

3   IMPLEMENTATION

Three aspects of an optimization were discussed in the methods section:
1.   Generation of the finite element model

2. Assigning parametric boundary conditions
3. Optimization algorithm and fitness to drive to an optimum

Each of these methods was implemented in order to automate the analysis and optimization of laminate composite parts made up of several lamina. The first method was implemented in a computer program to automatically generate the finite element model that can be read by commercially available finite element programs. The second method was implemented in a script that was written in a finite element package that automatically loads, constrains and solves the finite element model. The third method is implemented in a composite optimization structure.   This optimization takes the first two programs that were written and puts them into an optimization package called Isight. Isight enables these programs to be linked together in a sequence to perform the optimization. The following section will discuss the implementation of the methods in greater detail.

3.1   Automatic Generation of the Finite Element Model

The method that generates the finite element model was implemented to create a computer program. This program allows users to automatically create a finite element model for a laminate composite part. The computer program is written using the C++ programming language and uses a NURBS-based geometry library called GSNLib. The inputs to this program (described below) are a ply table, a material table, a mesh density and part geometry. The output is a composite finite element model which embodies the information from these input files.

The ply table is a spreadsheet that describes each lamina in the composite lay-up. Each line of the spreadsheet expresses that lamina's fiber angle, thickness, material, and priority. The priority describes how prevalent that lamina is used throughout the laminate. For example: a ply with a priority of 1 will likely be used more frequently than a ply with a priority of 7. An example of a ply table is shown in table 1. The material table is a spreadsheet that contains all the specific material properties for all of the composite materials that are used in the ply table. The mesh density is input as the number of elements in the u and v directions of the mid-surface of the part.

Ply	Priority	Material ID	Thickness	Angle	Material
1	1	1	0.1	0	graphite
2	3	1	0.1	30	graphite
3	5	2	0.1	60	glass
4	7	2	0.1	90	glass
5	6	2	0.1	-60	glass
6	4	1	0.1	-30	graphite
7	2	1	0.1	0	graphite

Tab. 1: An example of a ply table.

The part geometry is defined in two text files where each file contains the NURBS surface information necessary to create the opposing faces of the part. The reason the geometry input files are in a NURBS based neutral format as opposed to a commercial CAD package format is for increased speed during optimization. The increased speed comes because the optimization avoids having to start-up the CAD package for each iteration, thus eliminating the overhead of that operation. Despite the speed benefits, it is extremely difficult and tedious for most users to create a NURBS based surface by hand, without the use of a CAD package. Therefore a computer program was written to automatically translate surface data from the CAD package into the NURBS based text file format. This greatly simplified the part geometry creation process while still utilizing the speed of the CAD independent approach while performing optimization.

After the inputs are read into the program, a mid-surface of the part is automatically created. The mid-surface is then automatically meshed with quadrilateral laminated shell elements. After that, following the procedure discussed in the methods section, each element is then associated with the corresponding plies that make up the 3-dimensional component of the shell element. The result of the program is an analysis data file formatted for use by a wide variety of commercial Finite Element processing packages.

3.2   Automatic Boundary Conditions of the Finite Element Model

ANSYS was one of the FEA packages that were used to demonstrate the program's effectiveness. Applying automatic boundary conditions to the ANSYS finite element model was implemented using the ANSYS Parametric Design Language (APDL). APDL is a scripting language that allows users to automate ANSYS with the use of do-loops, if/else statements, and other automated ANSYS commands. Scripts can be easily written to automatically load, constrain and solve analyses. An ADPL script was written to automatically set up a modal analysis on a cantilevered wing. This script tells ANSYS what analysis to be performed (in this case a modal analysis), automatically constrain one end of the wing in all 6 degrees of freedom and specifies which modes to output to a file. The following example is a script that automatically solves a modal analysis of a cantilevered model with a 20x20 mesh density as shown in figure 2:

```
/solu
ANTYPE,2
MODOPT,SUBSP,10
EQSLV,FRONT
MXPAND,10, , ,0
LUMPM,0
PSTRES,0
MODOPT,SUBSP,10,0,0, ,OFF
RIGID,
SUBOPT,8,4,14,0,0,ALL

*SET,NodeNum,1
*SET, numInc,1
*SET,NumNodesU,20
*SET,NumNodesV,20
*DO, inc, 1, NumNodesV * NumNodesU, numInc
        d,NodeNum,ux,,,,,uy,uz,rotx,roty,rotz
        NodeNum = NodeNum + numInc
*ENDDO
SOLVE
FINISH

/POST1
*DO, i, 1, 10, 1
        *GET, mode, MODE, i,FREQ
        /OUTPUT,Modes,txt,,APPEND
        *VWRITE,i, mode
        Mode %I: %14.7G
        /OUTPUT
        /show,close
*ENDDO
```
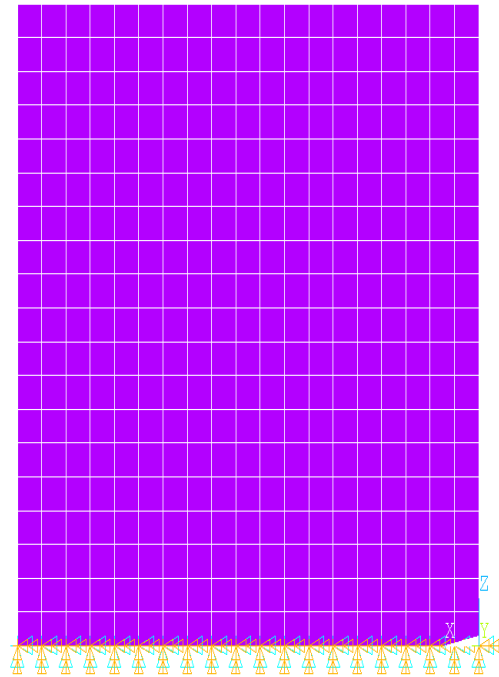

Fig. 2: An ANSYS shell model of a cantilever beam.

3.3   Optimization using Isight

A laminate composite optimization was set up using commercial optimization and process flow software called Isight by SIMULIA. The objective of the optimization is to drive the natural frequencies of the wing away from known keep out frequencies in order to avoid resonant conditions. The design

variables of the optimization are the fiber angles of each of the plies in the lay-up. These design variables are put into the Isight software along with the objective. A genetic algorithm is configured to drive the optimum away from the keep out frequencies by adjusting the fiber angles of each ply. The genetic algorithm first assigns random values from -90 to 90 degrees to the fiber angles. Over time the genetic algorithm evolutionarily drives these angles to values that make the wing's natural frequencies as far from the keep out frequencies as possible.

The fitness factor of this optimization is somewhat more complex than just the output from the analysis. Each modal frequency needs to stay as far away from the keep out frequencies as possible. Often when a modal frequency moves away from one frequency range, it starts to get closer to another keep-out frequency. The compromise in this case is to center the natural frequency between the two keep-out frequencies. To do this, the algorithm to determine the fitness first finds the distance between each natural frequency and the keep-out frequency. The algorithm then finds the difference between each natural frequency and its closest keep-out frequency. The fitness factor becomes the minimum of all these. The objective of the optimization is to maximize the minimum of all the minimum distances. The result is the optimum design's modal frequencies will be as far away to any of the keep out frequencies as possible.
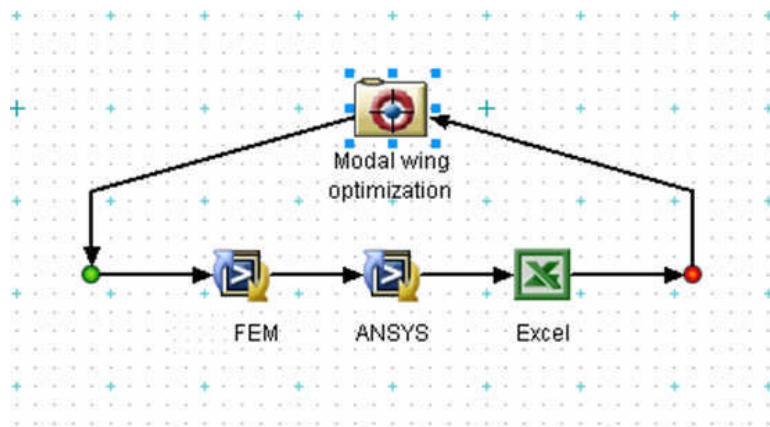

Fig. 3: iSight-FD process flow for the laminate composite optimization.

Figure 3 shows the process flow for the modal analysis optimization of the laminate composite wing described above. The modal optimization loop is run as many times as is needed to get the optimization to converge (possibly hundreds or even thousands of times). The first task in the optimization runs the code to set up the mesh and laminate properties of the finite element model. The next task runs ANSYS with the APDL script to automatically set up the constraints, solve the model and output results to a text file. In the third step, iSight takes the results from the text file and puts them into Excel. Excel is pre-programmed to determine the goodness of fit of the solution from the results of the optimization. The fitness result is then automatically taken from Excel and put back into the iSight optimization.

4   RESULTS

The automated composite finite element tool has been successful in automatically creating laminate composite finite element models for ANSYS, NASTRAN and LS-DYNA. Figure 4 shows three models of a wing. The first model is a wing created in NX. The second is a mid-surface shell element model of the wing created by the automated composite finite element tool and loaded into ANSYS. The model has a mesh density of 15x30 and contains thirteen, 0.1" lamina at the thickest area. The third model is the same ANSYS finite element model as the second, but it has the lamina thicknesses of the shell elements shown in 3-dimensions for better visualization.
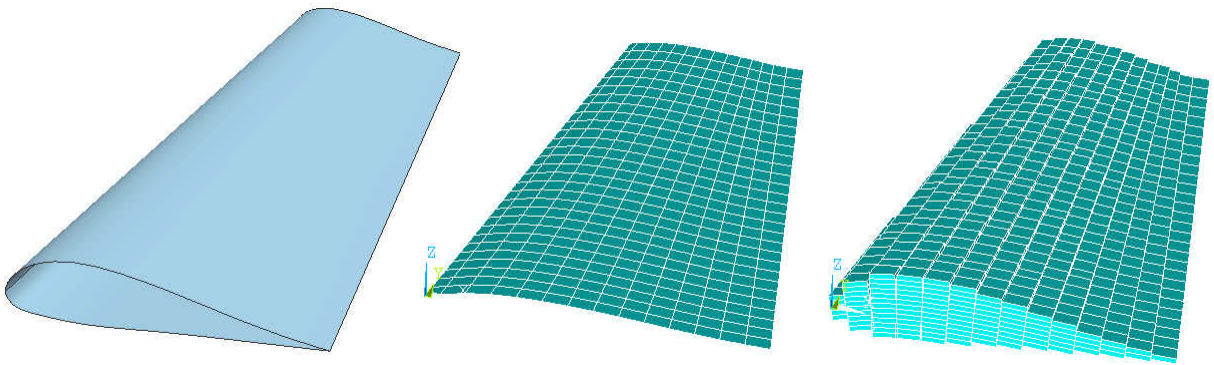
Fig. 4: Left: wing in NX. Middle: FE shell mesh. Right: FE shell mesh with lamina thicknesses shown.

The major advantage of using the automated composite finite element tool over other zone based, automated methods is that the finite element models that it creates are more detailed and thus will more accurately represent the laminate lay-up for models with large variation in surface topology. As shown in the far right image of figure 4, each element contains its own ply lay-up. Each ply in this lay-up is defined with some thickness (shown in figure 4), fiber angle (not shown) and material properties (also not shown). Because each element has its own ply lay-up, the finite element models can represent areas with large variation with more detail than those that are divided into a just a few zones as done in previous methods. Except for the case where the number of zones matches the number of elements in a finite element model, models with large variation in surface topology (possibly as simple as the one shown in figure 4 or as complex as figure 5) will be more accurately represented than the models created using a zone based method.

Figure 5 shows an example of a more complex topology that was modeled using the automated composite finite element tool. The mesh density is 40x40 nodes and the model contains 36 plies that are .01" thick. Although this example is not an actual part, it is useful to show here to demonstrate the complexity of models that can be made using this tool. Other zone based methods would not come close to being able to model this part with as much detail.
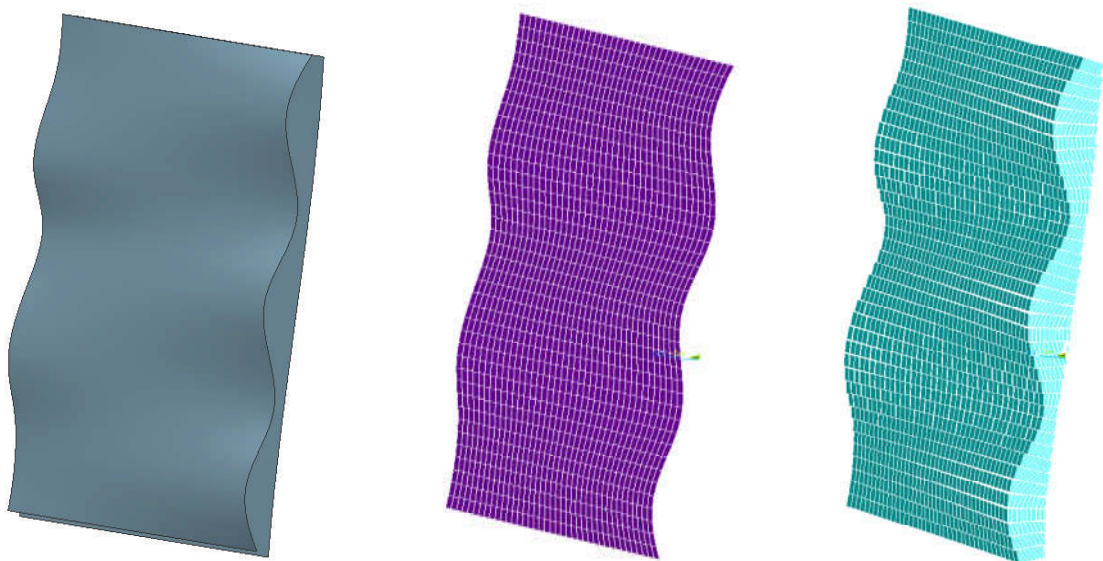

Fig. 5: Left: Complex NX Surface. Middle: FE shell mesh. Right: FE mesh with thicknesses shown.

There is also an obvious time savings advantage of using this automated approach instead of creating a laminate composite model directly in the FEA package. If a finite element model such as the one shown in figure 4 were done manually, it would take a proficient user an estimated 20-40 hours of work. This is because each of the 450 elements in that model needs to have a unique ply lay-up assigned to it. Using the automated composite finite element tool, the same model takes only a few seconds to create.

Table 2 shows run times of the automated composite finite element tool to create the wing model shown in figure 4. The run time increases approximately linearly with the number of total nodes in the model. As the number of plies in the model increases, the time increases by around 50%. This study shows that even finely meshes models with large numbers of plies can be created extremely quickly (under 30 seconds) as compared to manual methods.

Num plies	10x20 nodes	15x30 nodes	30x60 nodes	50x100 nodes
13 plies	0.281 sec.	0.687 sec.	2.766 sec.	7.609 sec.
26 plies	0.328 sec.	0.781 sec.	3.172 sec.	8.562 sec.
52 plies	0.422 sec.	0.969 sec.	4.047 sec.	10.844 sec.
104 plies	0.609 sec.	1.375 sec.	5.563 sec.	15.281 sec.
208 plies	0.906 sec.	2.171 sec.	8.67 sec.	24.172 sec.

Tab. 2: Speed results of the automated composite finite element tool.

The modal analysis optimization of a laminate composite wing as discussed in section 3.3 was successful in driving the modal frequencies away from the keep out frequencies. In this optimization the fiber angles of each of the 13 plies were used as the design variables and were free to move between -90 and 90 degrees. The genetic algorithm evolutionarily drove these angles to values that resulted in a maximum distance of the closest modal frequency to a keep out frequency. The result was that the first ten modal frequencies of the wing were at least 100 Hz away from any of the ten keep out frequencies. Figure 6 shows the progression of this optimization as it drove to an optimum design.
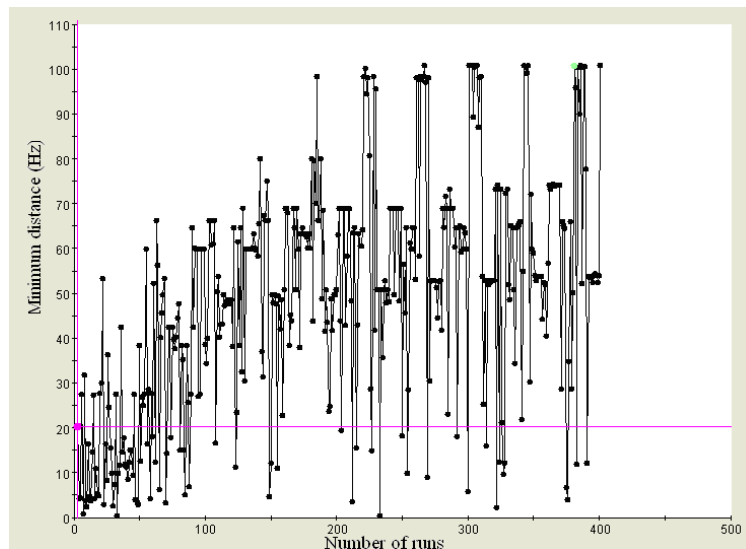

Fig. 6: A graph of the genetic algorithm's progression that drove the wing's natural frequencies away from the keep out frequencies.

5   CONCLUSIONS

The automated composite finite element tool presented in this paper creates laminate composite finite element models that are more detailed than those made with zone based methods. Because of the greater detail in the model, this method will likely yield more accurate analytical results for models with large variation in surface topology compared with similar models created with zones based methods. Continued research is needed to prove this through physical testing. Another advantage of this tool is that it automates the creation of laminate composite finite element models and will save engineers, who are doing this manually, dozens of hours of work per model. The automated composite finite element tool can also be used in conjunction with a method to automatically apply boundary conditions to create an effective optimization of a laminate composite part.

REFERENCES

[1]   Farin, G.: Curves and Surfaces for Computer Aided Geometric Design, Academic Press, Inc., San Diego, CA, 1988.
[2]   Gibson, R. F.: Principles of Composite Material Mechanics, CRC Press, Boca Rotan, FL, 2007.
[3]   Gurdal, Z.; Haftka, H.; Hajela, P.: Design and optimization of laminated composite materials, John Wiley & Sons, Canada, 1999.
[4]   Hale, R.; Schueler, K.: Knowledge-Based Software Systems for Composite Design, Analysis and Manufacturing, Society of Automotive Engineers, 2001.
[5]   Hepworth, A. I.; Jensen, C. G.; Roach, J. T.: A CAD Independent Approach to Automate Laminate Composite Design and Analysis, Computer-Aided Design & Applications, 6(2), 2009, 147-156.
[6]   Menayo, G.; Quero, A.: Computer-aided method of obtaining a ply model of a composite component. U.S. Patent App. 731,794, 2007.
[7]   Mitchell, Melanie: An Introduction to Genetic Algorithms, MIT Press, 1998.
[8]   Piegl, L.; Tiller, W.: The NURBS Book, Springer-Verlag, Berlin, Heidelberg, New York, 1997.
[9]   Rogers, D. F.; An Introduction to NURBS: with Historical Perspective, Academic Press, San Francisco, 2001.
[10]   Siemens Corp., http://www.plm.automation.siemens.com/en_us/Images/10651_tcm1023-4449.pdf, 2008.
[11]   Strong, A. B.: Fundamentals of Composites Manufacturing Materials, Methods, and Applications, Society of Manufacturing Engineers, USA, 2008.
[12]   Vasey-Glandon, V. M.; Kunkee, D.: Knowledge driven composite design optimization process and system therefore, U.S. Patent No. 7,010,472, 2006.
[13]   Weiss, P.: Long Island eyes growth in composite materials market, Long Island Business News, January 25, 2008.