

See discussions, stats, and author profiles for this publication at: <https://www.researchgate.net/publication/332766462>

Ansys Tutorial for ACP (Full composite tutorial in ANSYS)

Method · January 2017

DOI: 10.13140/RG.2.2.31473.53604

CITATIONS

0

READS

11,642

1 author:



Armin Yousefi Kanani

Lancaster University

11 PUBLICATIONS 3 CITATIONS

[SEE PROFILE](#)

Some of the authors of this publication are also working on these related projects:



Adhesive on multi material joints [View project](#)

ANSYS TUTORIAL

Modelling and Analysing Composite Model in
Ansys Software



Armin Yousefi Kanani

Mechanical Engineer

Email just for business inquiries:
a.yousefi.kanani@gmail.com

ArminYK

I am a Mechanical Engineer with over 3 years' professional experiences. I finished my Master degree in the field of Mechanical Engineering at Kingston University (London) in 2016 and gained a distinction. I completed my Mechanical Engineering BSc at Azad University of Qazvin-Iran in 2013.

I chose the subject of my MSc dissertation in a way that enabled me to work in experimental research and finite element analysis. Utilising all my experience and software expertise encouraged me to continue my research and dissertation in the field of composites. In my MSc project, critical buckling loads of laminated composite plates with different stacking sequences are determined through experimental and finite element method. The results from these two methods are compared and recommendation for replacing tests with modelling was made.

I have made these tutorials based on my knowledge that I got from my dissertation. This tutorial contains three main sections. The first section is dedicated to beginner who has never worked with Ansys software. In this section you will learn how to work with ANSYS software and also creating and analyzing simple composite model. In the second part of this tutorial you will develop knowledge to do eigenvalue, harmonic response and modal analysis in the ANSYS and also you will learn how to make sandwich composite model in ACP by using honeycomb core. In the last part of this tutorial, you will learn how to create complex geometry in ANSYS such as: lug, reinforced beam, pressure vessel, tensile test sample and modeling of delamination. Then you will learn how to do complex analysis by using more than one tools in project schematic such as delamination, tensile test, fatigue test, progressive damage and finding natural frequencies and mode shape of wing.

I hope you enjoy it
Aykanani



Armin Yousefi Kanani
Mechanical Engineer

@armin2020

For Business Enquirers: a.yousefi.kanani@gmail.com

Contents

1- ACP introduction (Modeling simple composite plate):	3
2- Creating Shell Pipe with Composite Material:.....	4
3- Draping in Composite Model.....	5
4- Using Rules Tools in ACP	6
5- Modelling Composite T-beam in ACP	7
6- Stress Analysis of Composite T-beam	8
7- Eigenvalue Simulation on simple composite plate.....	9
8- Natural Frequencies and Mode Shape of Composite Plate with Hole	10
9- Determining the harmonic response of a fixed-fixed Composite Beam	11
10- Creating Shell Honeycomb model by Using Honeycombcreator Extension.....	12
12- Stress Analysis of Pressure Vessel (Solid modeling):.....	14
13- Reinforced Beam with Combination of Steel and Composite Material.....	15
14- Find the Best Laminate Configuration of a Composite Tube under compressive load:	
15- Modeling Lug in ACP (complex geometry):	17
16- Tensile Test:	18
17- Determining the Fatigue Behavior of Composite Tensile Specimen:	19
18- Finding 5 Natural Frequencies and Mode Shape for Pre-stressed Composite Wing.....	
19- Investigating Mode I with Interface Elements (Delamination):.....	21
20- Modeling Progressive Damage for Composite Plate:.....	22

Future works:

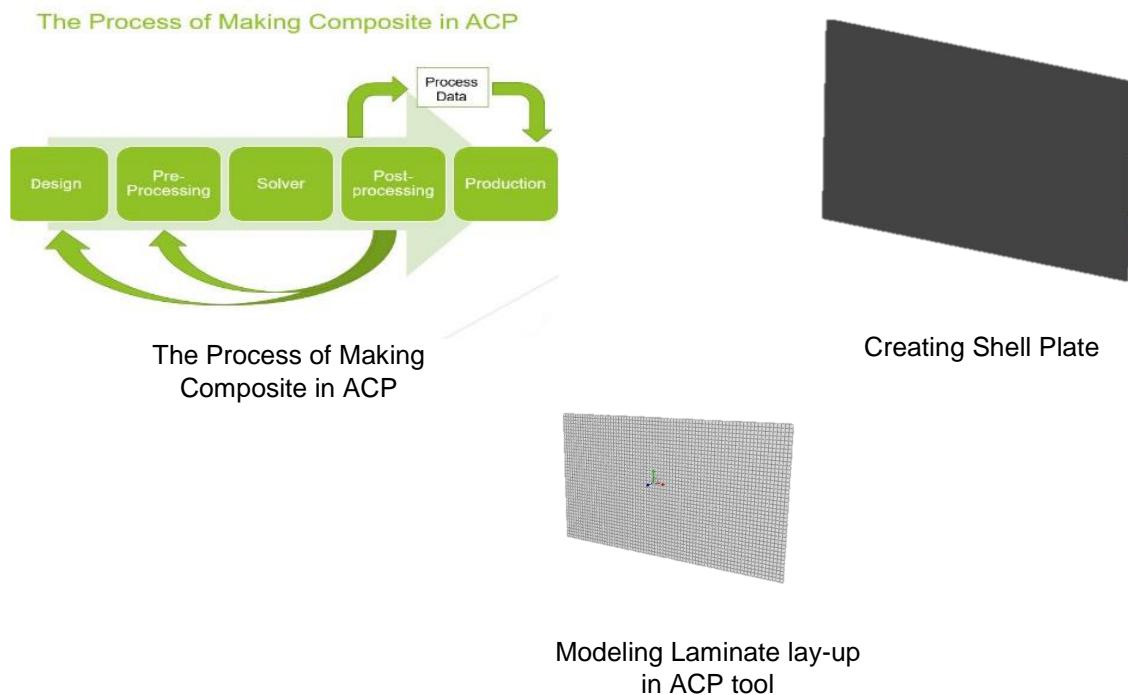
- 1- Debonding between composite layers
- 2- Single lap joint
- 3- Composite tube under quasi-load by using ls-dyna
- 4- Impact test of composite material by using ls-dyna
- 5- Bird strike on composite wing
- 6- Crash test on composite model
- 7- Creating rotor model with composite material in ACP
- 8- Cure simulation of composite material
- 9- Using n-code life tools with ACP in ANSYS

Introduction

Composite materials are created by combining two or more layered materials, each with different properties. These materials have become a standard for products that are both light and strong. Composites provide enough flexibility so products with complex shapes, such as boat hulls and surfboards, can be easily manufactured. Engineering layered composites involves complex definitions that include numerous layers, materials, thicknesses and orientations. The engineering challenge is to predict how well the finished product will perform under real-world working conditions. This involves considering stresses and deformations as well as a range of failure criteria. ANSYS Composite PrepPost provides all necessary functionalities for the analysis of layered composite structures.

1- ACP introduction (Modeling simple composite plate): Level: Amateur

In this tutorial you will have introduction of Ansys software (Toolbox, graphic view, command, etc.) You will learn who to add tools into project schematic and how to use multiple tools and also the meaning of connector in Ansys. In the next step, you will have full introduction of ACP tools which is used to model composite model in Ansys software and its usages and applications. In this tutorial we are going to model simple composite plate. First, you will learn who to add composite material from Ansys library or creating new composite material in engineering data section. Then you will learn who to create shell model in geometry and apply mesh in model section and at the end we are going to apply composite laminated lay-up on our shell model by using ACP Tools.



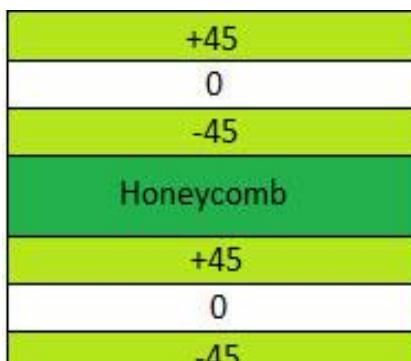
2- Creating Shell Pipe with Composite Material:

Level: Amateur

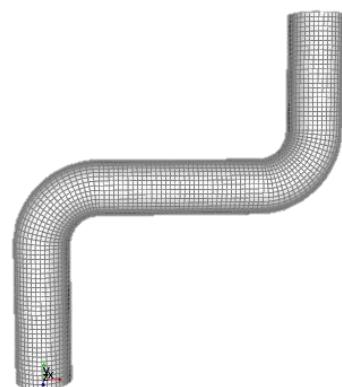
Rosettes are coordinate systems that used to set the Reference Direction of Oriented Selection Sets. Rosettes define the 0° direction for the composite lay-up. Coordinate systems defined in Mechanical are imported by default. Additional Rosettes can be defined in ACP. There are 5 different types of Rosettes. The origin and directions of Rosettes are given in the global coordinate system. This Tutorial will help using edges to orient fiber directions. Very often fiber directions follow the outer structure of a design. To use this when defining fiber directions ANSYS Composite PrepPost can define the directions based on edges in the model.

Agenda:

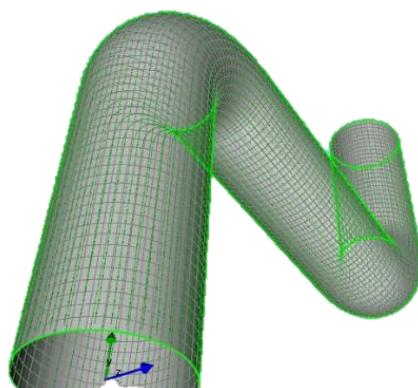
- Define Stackups
- Define Edge Sets
- Use Edge Wise Rosettes to define Reference Directions



Laminated Lay-Up



Pipe Geometry



In ACP you can see fiber direction on each layer

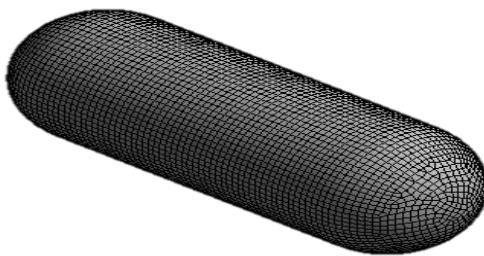
3- Draping in Composite Model

Level: Amateur

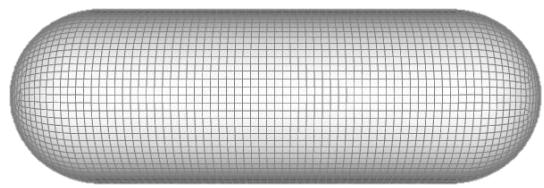
The ply application (draping) on doubly curved surfaces changes the theoretical fiber orientations. In many cases the effect is small and can be neglected. On the other side it is important to know how big this effect can be and if it has to be considered. ACP allows you to evaluate the draped fiber directions. In this tutorial we are going to model pressure vessel in design modeler in ANSYS software. Then we will apply composite layers on shell model. In the next step we are using different technique to analyzing draping in software.

Agenda

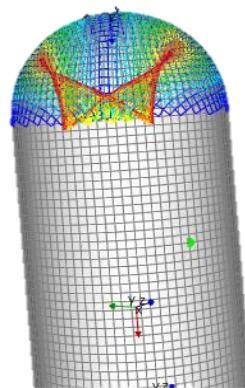
- Define draping for modeling groups
- Select a seed point
- Show draped fiber directions
- Show draping shear
- Show and export flatwrap
- Define draping based on an oriented selection set
- The effect of different seed points



Modeling Pressure Vessel in Design Modeler



Applying Composite layers on Pressure vessel



Overlapping on Composite Layers

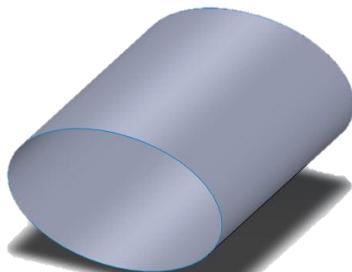
4- Using Rules Tools in ACP

Level: Amateur

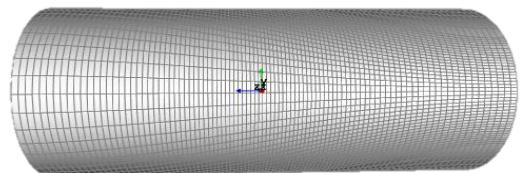
Selection Rules allow you to select elements through geometrical operations. These selections can be combined with Oriented Selection Sets or Modeling Plies to define plies of arbitrary shape. This feature can be used to define local reinforcements (patches) or staggering. In this tutorial, first we are modeling tube geometry in CAD software then we will import PARASOLID format file into ANSYS geometry. Then we will apply mesh and create name selection. In the last step in ACP we are going to have introduction of rules command in ACP and we will see its usage and application in the software.

Agenda

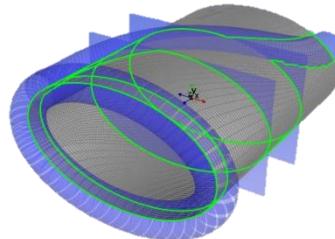
- Introduction to rules
- Using Parallel rules
- Using a tube rule following an edge
- Using a tube rule and parallel rule at same time
- Using a template rule



Modeling tube geometry in
CAD software



Importing Tube geometry into ACP
and then applying composite layers



Using rules command in ACP Tools to
apply composite layers on specific area

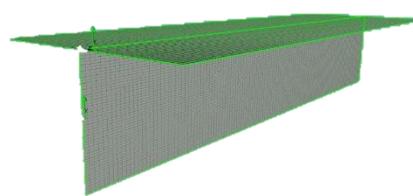
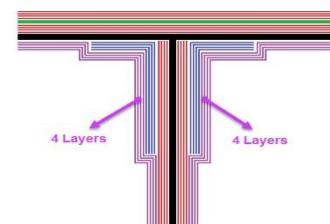
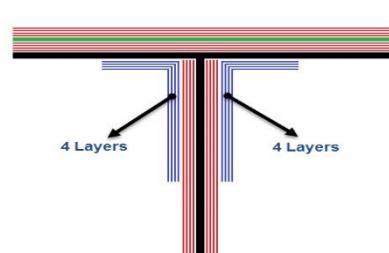
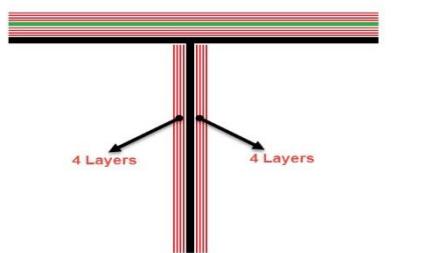
5- Modelling Composite T-beam in ACP

Level: Amateur +

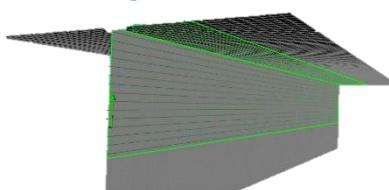
An Oriented Selection Set is an Element Set with additional information about the element orientations. The orientation direction of an Element Set is responsible for setting the stacking direction of the associated lay-up. The Reference Direction is responsible for setting the 0 direction of the associated layup. These two directions must be defined while other parameters like Selection Rules and Draping are optional. The laminate is modeled by defining different Oriented Selection Sets for the different regions. The modeling plies are then associated with the OSS and their order defines the stacking sequence of the laminate. The OSS feature allows you to define several offset directions for one element: OSS can overlap and can have different orientations. In this tutorial we are going to model T-Beam by defining various oriented selection sets. This beam is made from three main parts which are: Top and middle surface, Bonding and covering layers.

Agenda:

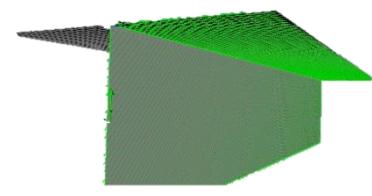
- Using multiple oriented selection sets to define a complex composite layup
- Tapering of core materials
- Using multiple rosettes and different selection methods



Applying Layers on top and middle surfaces



Applying bonding layers



Applying covering layers

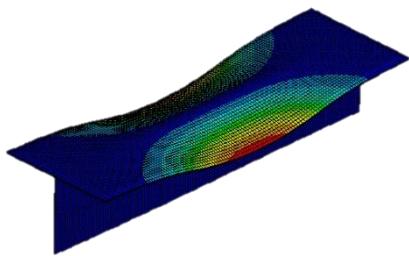
6- Stress Analysis of Composite T-beam

Level: Intermediate

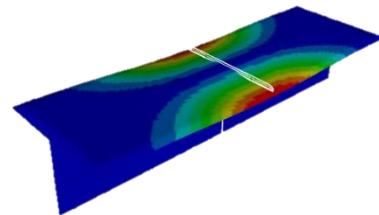
ACP has a pre- and post-processing mode. In the pre-processing mode, all composite definitions can be created and are mapped to the geometry (FE mesh). These composite definitions are transferred to the FE model and the solver input file. In the post-processing mode, after a completed solution and the import of the result file(s), post-processing results (failure, safety, strains and stresses) can be evaluated and visualized. After a completed solution, the post-processing is used to evaluate the performance of the design and laminate. The post-processing functionality for Solid Models of ACP allows the mapping of ply wise results on to the reference surface of the solid model. This ensures that also failures occurring inside the laminate can be observed and investigated. Failure criteria are used to evaluate the strength of a composite structure. Several failure criteria can be defined, combined and configured in the Definitions object. Failure criteria definitions can be used for failure plots and sampling points. The critical failure mode for an element shown in failure plots and sampling points is the one with the lowest reserve factor.

Agenda

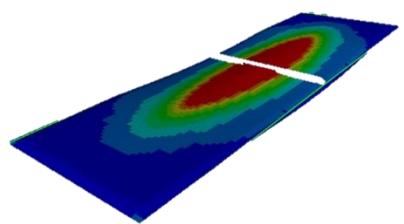
- Connecting ACP to Static Structural Tools in Ansys
- Doing Stress Analysis
- Solving the model
- Analyzing Data in static Sturcutal
- Analyzing Data in Post-Processing



Total Deformation in Static



Total Deformation in Post ACP



Failure in Top Surface of T-beam
in Post ACP

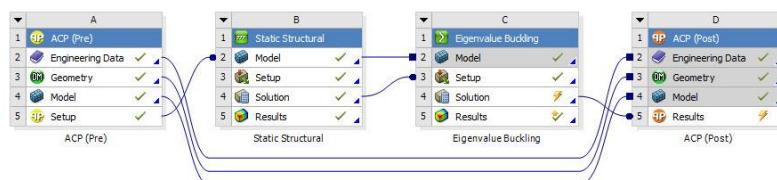
7- Eigenvalue Simulation on simple composite plate

Level: Intermediate

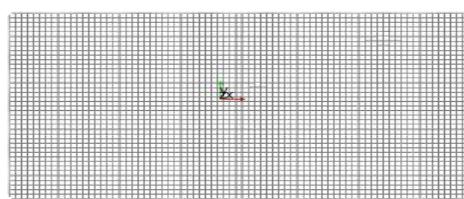
Liner buckling (also noun as eigenvalue buckling) analysis estimated conceptual buckling strength of an optimal elastic structure. The eigenvalue can be calculated by the given constraints and system loading. In real-life nonlinear solution is utilized to obtain most accurate results- in comparison to linear solution. Thereby, this method is not suggested for precise buckling estimation. Moreover, some buckling problems just need to be solved with non-linear solution because it cannot be estimated with linear solution. Analyses that require upstream results from Eigenvalue Buckling or Pre-Stress Modal analyses are also supported and can be transferred from the solution.

7-1 Supported Mechanical Systems for ACP models

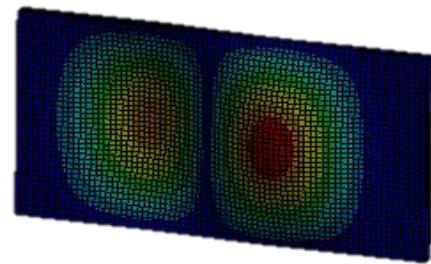
- Static Structural
- Transient Structural
- Steady-State Thermal
- Transient Thermal
- Modal
- Harmonic Response
- Random Vibration
- Response Spectrum
- Eigenvalue Buckling



The Process of Buckling Simulation



Creating simple plate in ACP



First Mode Shape

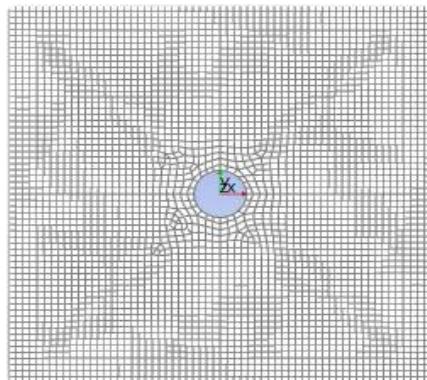
8- Natural Frequencies and Mode Shape of Composite Plate with Hole

Level: Intermediate

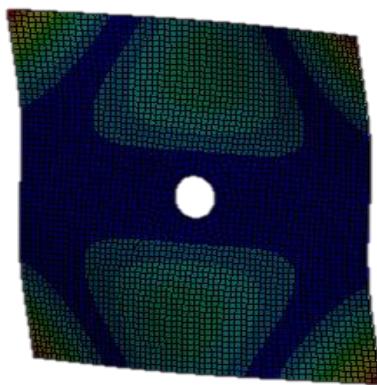
The modal analysis technique is used to determine the vibration characteristics (I,e, Natural frequencies and mode shapes) of linear elastic structures. Our goal is to determine the first 10 natural frequencies and mode shapes for the plate with the hole shown. In this tutorial, first we are going to model simple plate with hole in ACP tools then we are importing model into modal. In the next step, we will have short introduction to modal tools in ANSYS software and do simulation to find natural frequencies and mode shape of composite model.

Agenda:

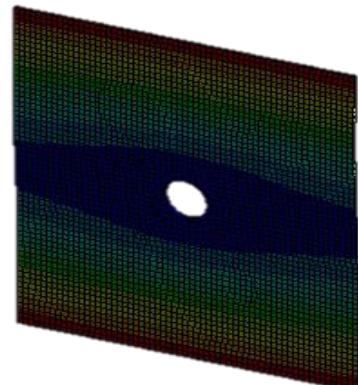
- Creating Plate with Hole in ACP
- Importing ACP model into modal analysis
- Applying boundary conditions
- Finding mode shapes



Creating Composite model in ACP



Mode 10 Shape in modal



Mode 1 shape in modal

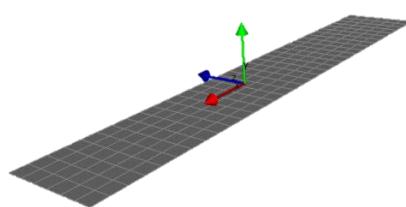
9- Determining the harmonic response of a fixed-fixed Composite Beam

Level: Intermediate

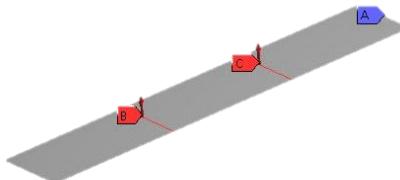
A harmonic analysis is used to determine the response of the structure under a steady-state sinusoidal (harmonic) loading at a given frequency. A harmonic, or frequency-response, analysis considers loading at one frequency only. Loads may be out-of-phase with one another, but the excitation is at a known frequency. This procedure is not used for an arbitrary transient load. vibration analysis should be always run prior to a harmonic analysis to obtain an understanding of the dynamic characteristics of the model. In this tutorial we are going to create composite beam then we will apply fixed-fixed boundary condition in modal section and find 5 mode shapes of beam. After finding mode shapes we will find harmonic response of beam under 250 N load applied on the two places in the middle of plate.

Agenda:

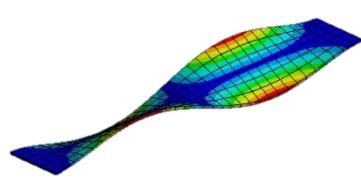
- Creating Composite beam in ACP
- Importing ACP model into modal analysis
- Finding 4 Mode Shape of Beam
- Determining the Harmonic Response
- Applying boundary conditions
- Finding mode shape



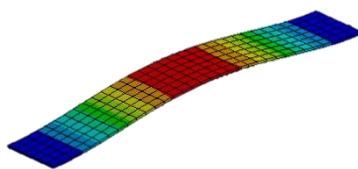
Modeling Beam In ACP



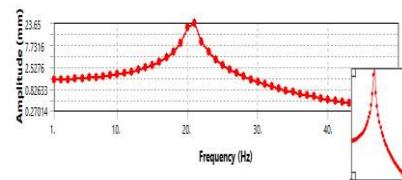
Applying Load and Boundary Condition



Mode 4 shape of beam



Harmonic response shape



Harmonic Response

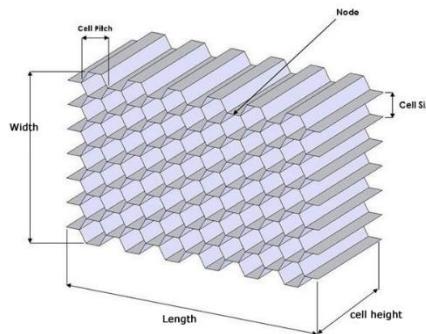
10- Creating Shell Honeycomb model by Using Honeycombcreator Extension

Level: Intermediate

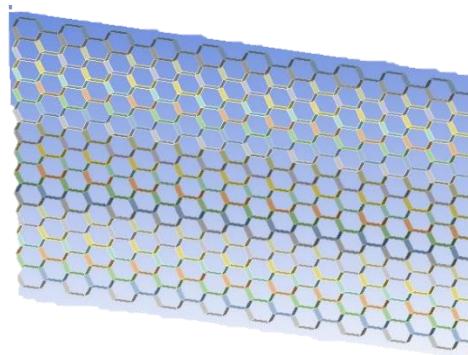
With ANSYS ACT, you can create a customized simulation environment that enables your engineering team to capture and use expert knowledge, specialized processes, and best practices, improving productivity, efficiency and effectiveness. The software enables you to encapsulate APDL scripts, create custom menus and buttons to incorporate your company's engineering knowledge, embed third-party applications, and create your own tools to manipulate simulation data. In This tutorial you will learn how to use ACT in ANSYS software. First, we need to download HONEYCOMBCREATOR extension from ANSYS application store. Then we will install extension. Then we will create shell model of honeycomb in design modeler.

Agenda:

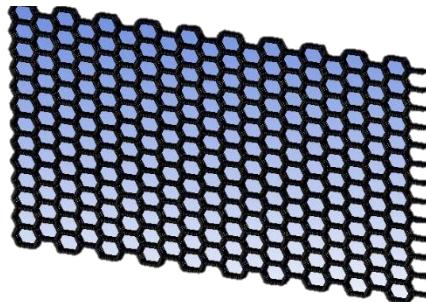
- Using ACT extension
- Downloading and installing extension in Ansys software
- Creating honeycomb by new extension



These are information you need to create your honeycomb



Creating Shell Honeycomb by using Honeycombcreator



Applying mesh on Honeycomb

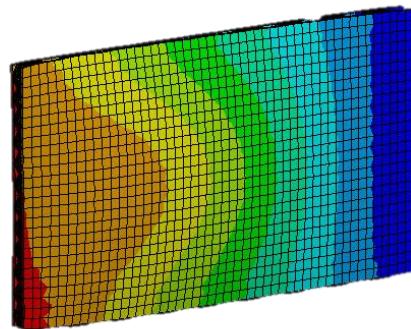
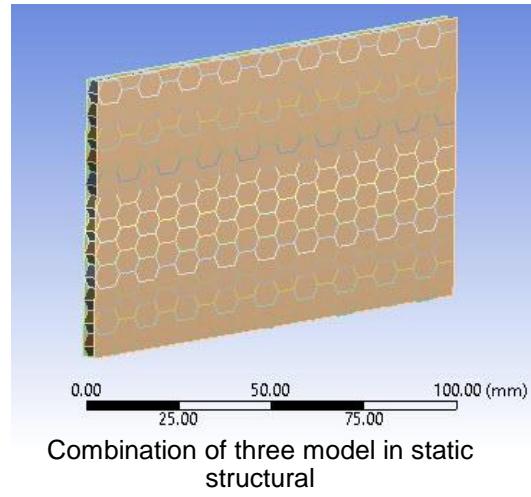
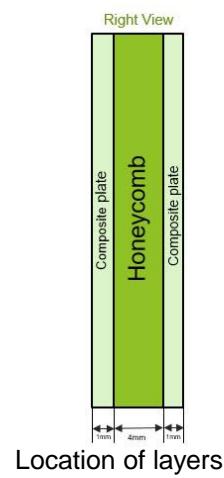
11- Creating Sandwich Composite Plate with Honeycomb Core

Level: Intermediate

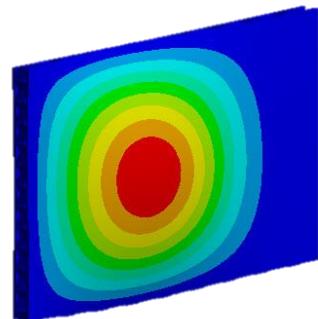
Honeycomb structures are natural or man-made structures that have the geometry of a honeycomb to allow the minimization of the amount of used material to reach minimal weight and minimal material cost. The geometry of honeycomb structures can vary widely but the common feature of all such structures is an array of hollow cells formed between thin vertical walls. In this tutorial we are going to create sandwich composite plate. First, we will create two composite plates in two different ACP and then we will create honeycomb in mechanical model by using Honeycombcreator extension. Then, we will combine three models into static structural to do stress analysis. At the end we will do eigenvalue simulation.

Agenda:

- Using two ACP to create two composite plates
- Creating Honeycomb by using honeycomb creator
- Apply boundary conditions, loads and solve the model



Total deformation of sandwich composite plate



First mode of buckling of composite sandwich plate

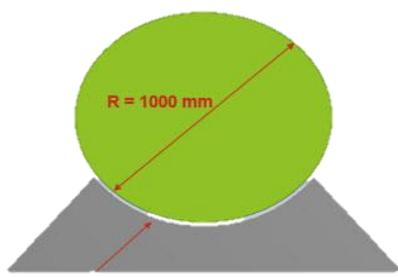
12- Stress Analysis of Pressure Vessel (Solid modeling):

Level: Advance

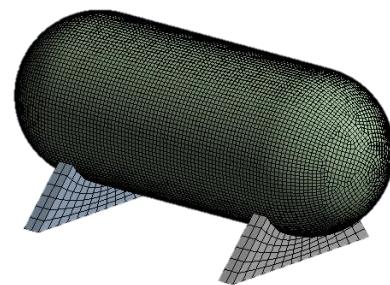
Meshes can be imported into Mechanical from multiple systems. It is possible to combine composite shell, composite solid, and non-layered shell and solid meshes to perform mixed analysis. Mechanical does not allow overlap of node/element number from multiple systems; therefore, the import fails if the meshes from different systems are overlapping in node/element numbers. Meshes from upstream to downstream Mechanical Models are renumbered automatically to avoid any overlap. For every analysis system (mesh), you have the choice between automatic renumbering (default) and manual configuration. If automatic renumbering is disabled, you must ensure that the element/node numbering is unique for each mesh. In this tutorial, first we will model Composite pressure vessel in ACP then we will model metal parts in mechanical model and at the end we will combine models in static structural.

Agenda:

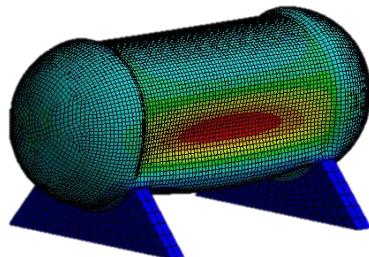
- Solid modeling of composite vessel
- Add metal parts
- Transfer parts to ANSYS Mechanical
- Apply boundary conditions, loads and solve the model
- Evaluate solid model results in ANSYS Composite PrepPost



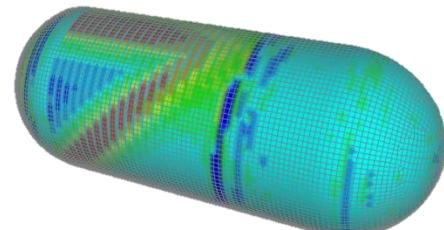
Geometry Dimensions



Assembled model in Static Structural



Total deformation in static structural



Total deformation on Post ACP on just composite part

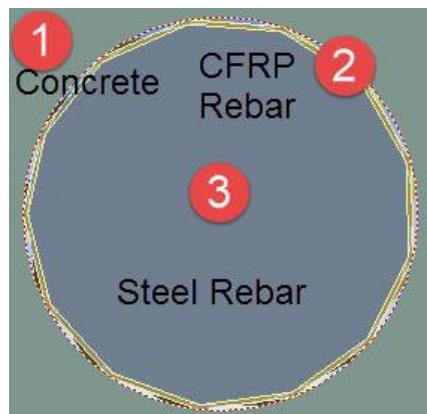
13- Reinforced Beam with Combination of Steel and Composite Material

Level: Advance

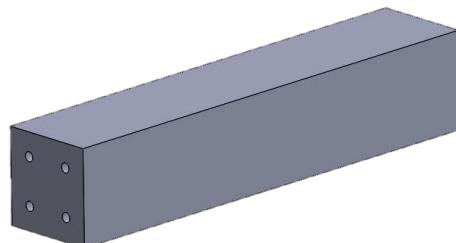
Reinforced Concrete is a common building material for construction of facilities and structures. While concrete has high compressive strength, it has limited tensile strength. To overcome these tensile limitations, reinforcing bars are used in the tension side of concrete structures. In this tutorial, first we are going to model concrete geometry in mechanical model then we need to model 4 structural steel rebar in another mechanical model. at the end we are going to reinforced structural steel rebar by using 4 composite layers

Agenda:

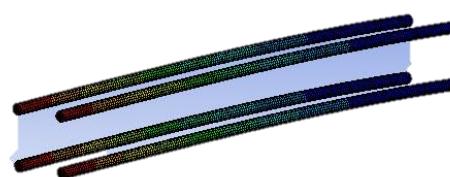
- Creating all models in CAD software
- Using three different type of materials
- Connecting two mechanical models and ACP into static structural
- Applying boundary conditions, loads and solve the model
- Applying variable pressure on beam



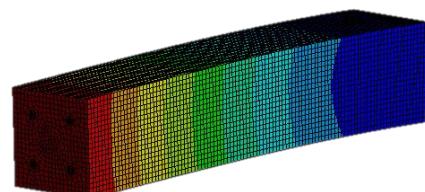
location of each material



Concrete beam geometry



Displacement on structural steel rebar reinforced by composite material



Displacement on beam reinforced by structural steel and composite material

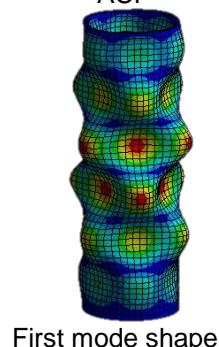
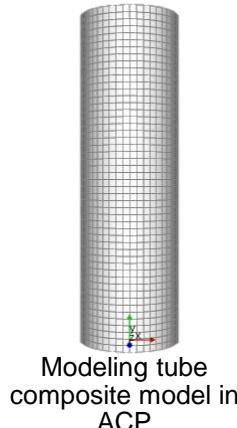
14-Find the Best Laminate Configuration of a Composite Tube under compressive load:

Level: Advance

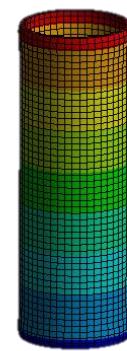
The Parameter feature connects Inputs and Outputs to the Parameter Interface in the Workbench project. There are three numerical variables which are diameter, laminate code and thickness. These variables should be applied to ascertain optimum condition for making composite. In this tutorial we are going to solve problem for specific laminated-lay up to find critical buckling load on the composite tube model then by using parameter command in ANSYS we will ask software to solve problem for various laminate lay-up, thickness and tube diameter.

Agenda:

- Modeling composite tube geometry in ACP tools
- Define input parameters for the ply angle of all eight defined plies
- Importing model into static structural and applying boundary conditions and force
- Solve the model and define output parameter for Eigenvalue amount
- Specify design variations in ANSYS Workbench and update all design points



First mode shape



Total Displacement

L	I	Z	D	E	r	G	R	J	K	N	S	U
I	10	nr	2									
I	P1	Y	0	1	0	(X	0	X	1.00000	0	/
I	P1	Y	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	-45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.00000	0	
I	P1	X	0	45	0)	X	0	X	1.00000	0	
I	P1	X	0	0	0	0	X	0	X	1.		

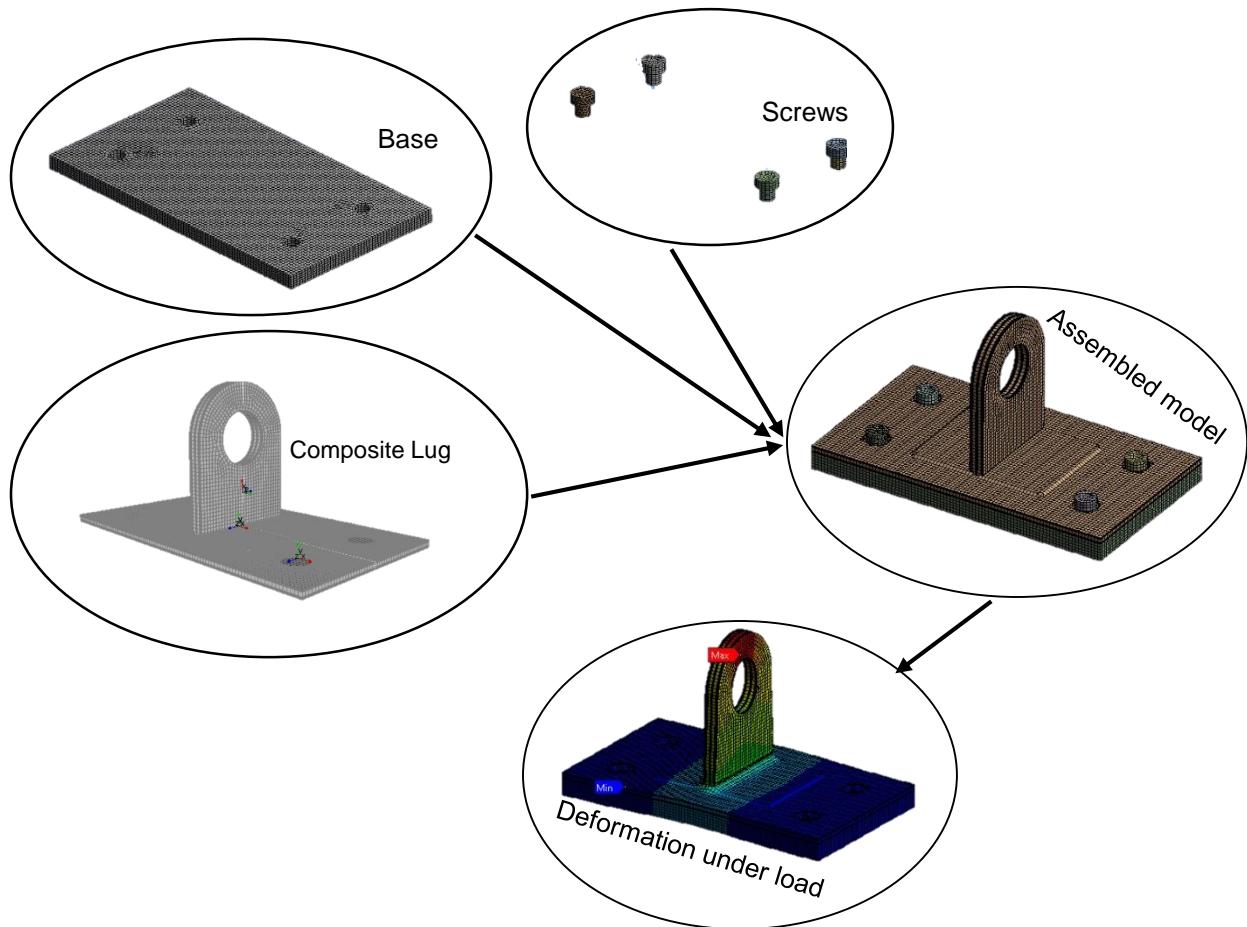
15- Modeling Lug in ACP (complex geometry):

Level: Advance

Solid model settings control how the solid model is divided into elements in the thickness direction and how drop- and cut-off elements are handled. The level of detail required in the solid model depends on how accurate certain features are to be modeled. This depends on the judgment of the designer. Extrusion guides add more complex possibilities in shaping the model. The cut-off operation is analogous to machining a composite part after curing. In this tutorial we are going to use extrusion guide, cut off element and snap to geometry to create complex geometry.

Agenda:

- Create complex composite model
- Import geometry for cut-off rule
- Creating Virtual Geometries
- Import geometry for Extrusion-guides feature
- Extrude solid composite using Extrusion- guides feature



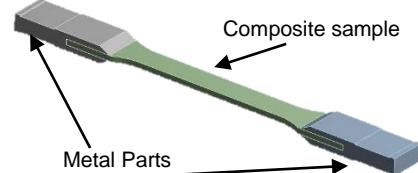
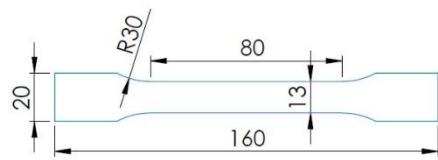
16- Tensile Test:

Level: Advance

Tensile testing, is also known as tension testing is a fundamental materials science test in which a sample is subjected to a controlled tension until failure. The results from the test are commonly used to select a material for an application, for quality control, and to predict how a material will react under other types of forces. In this tutorial, first we are going to model specimen geometry in CAD software then we will apply composite layers in ACP tools. In the next step, we are going to model two metal parts and then at the end we will combine models into static structural. Meshes can be imported into Mechanical from multiple systems. It is possible to combine composite shell, composite solid, and non-layered shell and solid meshes to perform mixed analysis.

Agenda:

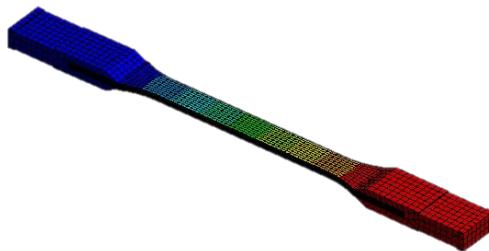
- Modeling solid composite specimen
- Add metal parts
- Transfer parts to ANSYS Mechanical
- Apply boundary conditions, loads and solve the model
- Evaluate solid model results in ANSYS Composite PrepPost



Composite specimens and two metal end tab



Applying mesh



Directional deformation of specimen on X direction

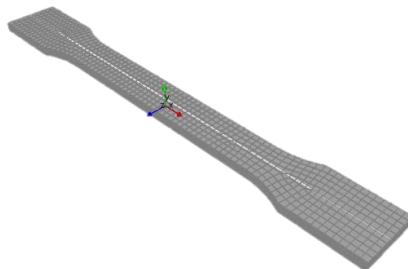
17- Determining the Fatigue Behavior of Composite Tensile Specimen:

Level: Advance

Random vibration analysis is another spectral method. The purpose of a random vibration analysis is to determine some statistical properties of a structural response, normally the standard deviation (1σ) of a displacement, force, or stress. (1σ) Is used to determine fatigue life of structure. In this tutorial, first we will model tensile specimen in ACP tools. Then we need to find 10 modes of specimen between 0 Hz to 1500 Hz frequencies. After that we need to calculate Sigma RMS, N, occurrence and V0 to calculate life time of specimen.

Agenda:

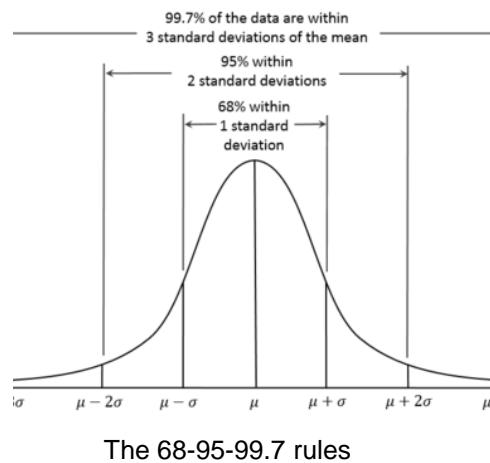
- Creating Composite Tensile specimen in ACP
- Importing ACP model into modal analysis
- Finding Mode Shapes
- Introduction of random vibration
- Applying boundary conditions
- Determining the fatigue behavior of composite tensile specimen



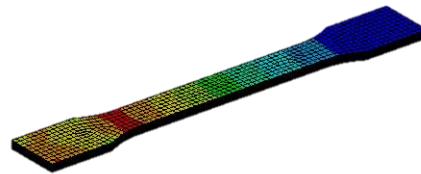
Solid modeling of Composite tensile specimen



The mode shape of composite tensile specimen



The 68-95-99.7 rules



Equivalent stress/sigma 1/
Probability: 68.269%

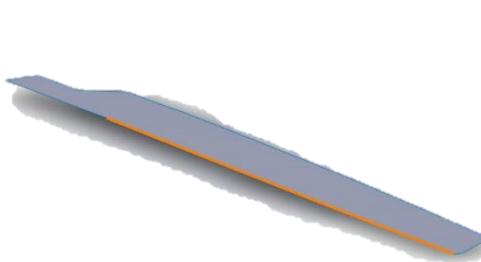
18-Finding 5 Natural Frequencies and Mode Shape for Pre-stressed Composite Wing

Level: Advance

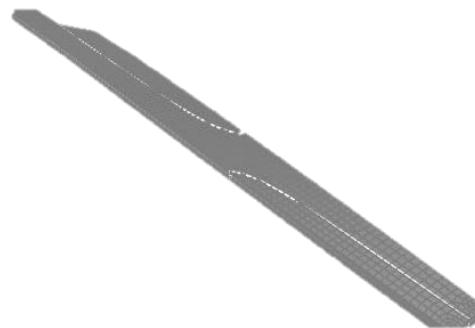
We use modal analysis to determine the natural frequencies and mode shapes of a structure. The natural frequencies and mode shapes are important parameters in the design of a structure for dynamic loading conditions. They are also required if you want to do a spectrum analysis or a mode superposition harmonic or transient analysis. In this tutorial we are going to find 5 natural frequencies and mode shape for pre-stressed solid composite wing.

Agenda:

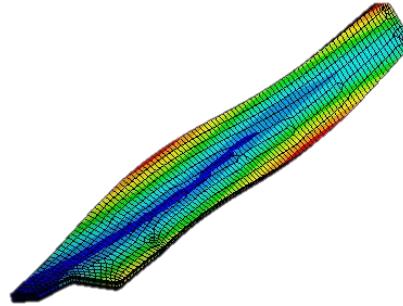
- Creating solid wing in ACP
- Applying boundary conditions
- Calculating pre-stress in static structural
- Finding natural frequencies and mode shapes



Modeling shell wing in CAD software



Creating solid wing in ACP



Total deformation on mode 5

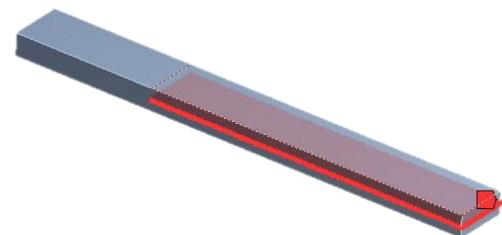
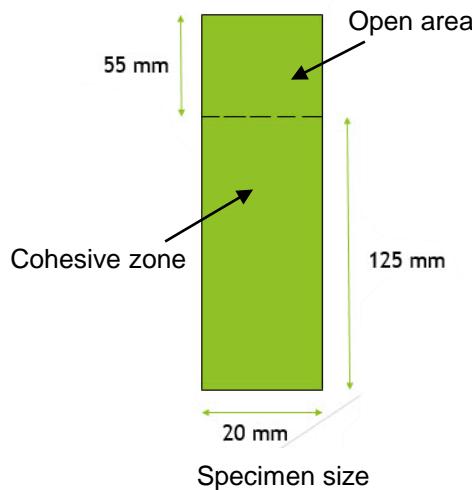
19- Investigating Mode I with Interface Elements (Delamination):

Level: Advance

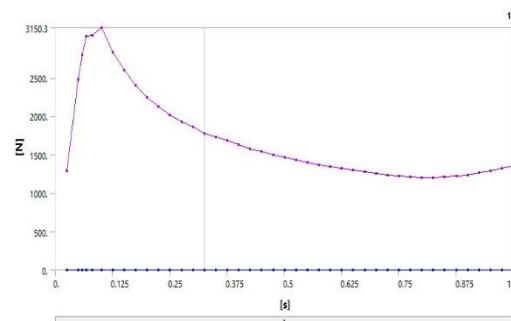
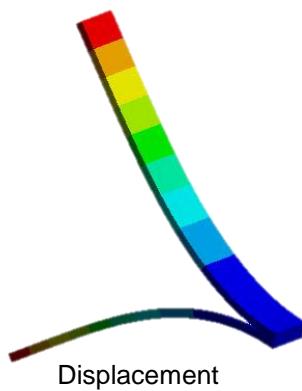
Composite fracture occurs at three modes, depending on three loading scenarios. Mode I is an opening or tensile mode, mode II fractures consist of a sliding or shear effect and mode III fractures occur due to a tearing or shear loading effect. This report will only be looking at fracture modes of type I and II. In practical occurrences of material fracture caused by cracking, especially in metals, mode I type of fracture is predominantly present.

Agenda:

- Set up properties for delaminating interfaces inside engineering data
- define delaminating interfaces inside ACP
- Use predefined interfaces from ACP inside Mechanical
- Alternatively use contact definitions with cohesive zone material inside Mechanical
- Boundary conditions and analysis settings
- Postprocessing



Defining cohesive zone and also interface delamination



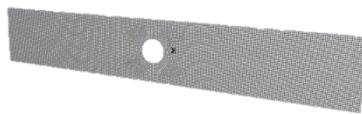
Force-Time Graph

20- Modeling Progressive Damage for Composite Plate: Level: Advance

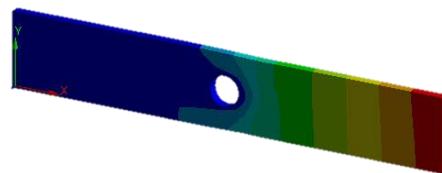
The progressive damage plot object can display progressive damage results. The overall damage status can be displayed indicating whether no damage, some damage, or complete damage has occurred. The damage variables give an indication of how much the stiffness has reduced for fiber and matrix in tension or compression. The damage variable scale goes from 0 (no damage, 0% stiffness reduction) to the maximum stiffness reduction specified. The highest reduction possible is 1 (complete damage, 100% stiffness reduction).

Agenda:

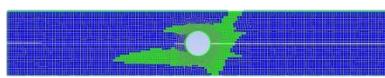
- Defining the engineering properties
- Defining the damage initiation and damage variable
- Setup composite lay-up in ACP (Pre)
- Apply boundary conditions, loads and solve the model
- Post processing



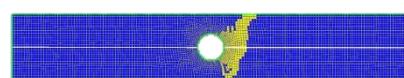
Modeling shell composite model in ACP



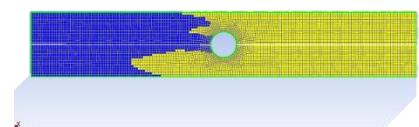
Total deformation under tensile force



Damage status in first ply



Fiber tensile damage on first ply



Matrix tensile damage on second ply

Thank You

Success is nothing more than a few simple disciplines, practiced every day

Jim Rohn