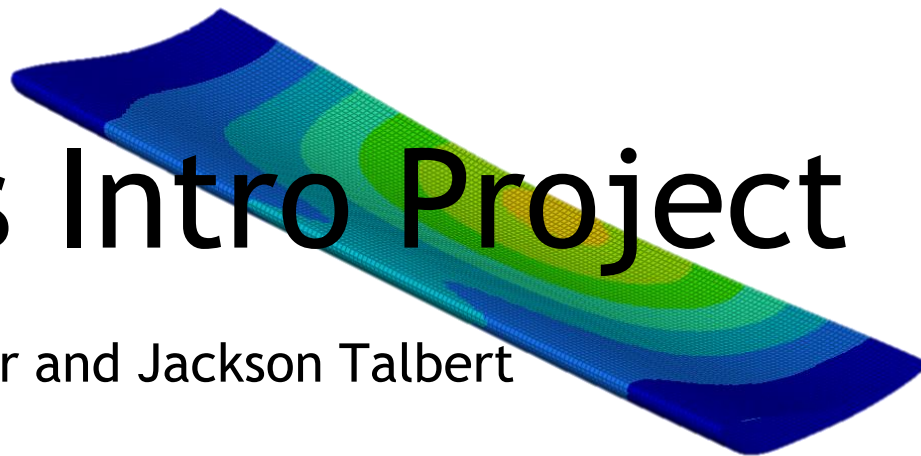




Structures Intro Project

Nick Garber and Jackson Talbert





Welcome + Foreword

Great, you finished flipping through our intro FEA presentation! Now, let's take that knowledge and apply it to a part on our car.

This project will reference info from those slides and build upon it. If you have not yet gone through them, it is strongly recommended. Take notes even. Use them as an aid and test your knowledge as you work through this.

There will be green boxes scattered that ask you questions and should get you thinking about the methodology used for this simulations. You don't need to do anything with them, but they will really help ***build your understanding***. Reach out to either of us if you have questions or want to discuss them.



Welcome + Foreword

This project will take a few hours and can be done in multiple sittings, no rush. After you get some experience, the project can be completed from start-to-finish in under 30 minutes. After completion of the initial project, you can continue to do the challenge if you'd like.

While we do our best to really spell everything out, google things that you don't understand or want to get more information on. Reach out to us if you need help!

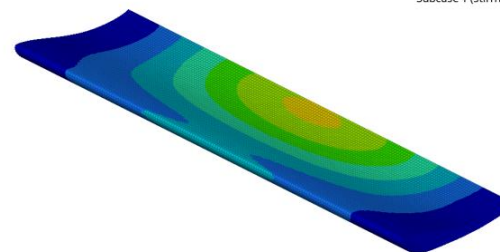
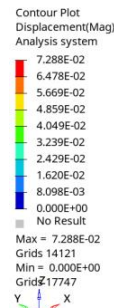
Without further ado, let's introduce the project...



Project Intro

For this intro project, you will be analyzing a front wing element under maximum aerodynamic loads. You will be walked through:

1. Geometry import
2. Meshing
3. Material & property card creation
4. Boundary condition setup
5. Linear static analysis setup
6. Post processing



1: 1
Subcase 1 (stiffness) : Static Analysis : Frame 0

Once the initial project is complete, we strongly encourage you to give the challenge a try!



Simulation Basics

This slide deck will run you through an example FEA simulation of a front wing element.

Planning the analysis

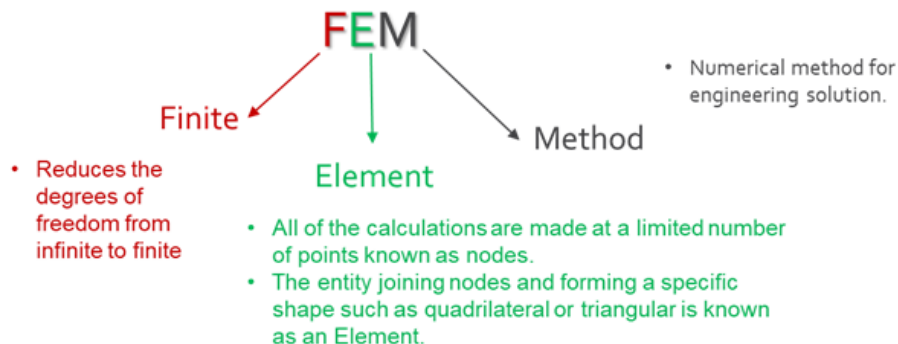
- What information are you trying to get from this model and how will you get it?

Discretization

- How would you like simplify this model to make it solvable?

Boundary Conditions

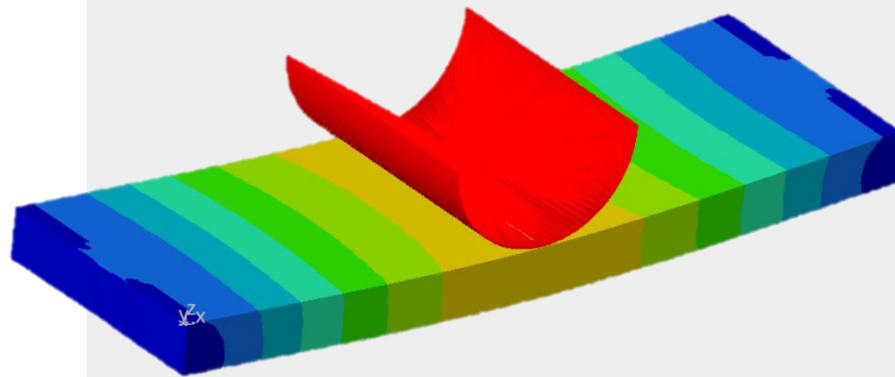
- How will you model the environment that this part operates within?





Planning The Analysis

What are your goals?



3 Point Bend Simulation

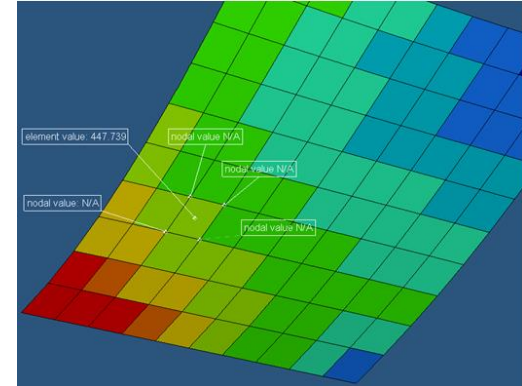


Planning The Analysis: Solution Type

The objective of this project is to determine the composite layup for a front wing element under maximum aerodynamic loading conditions.

Given this statement, we know the following:

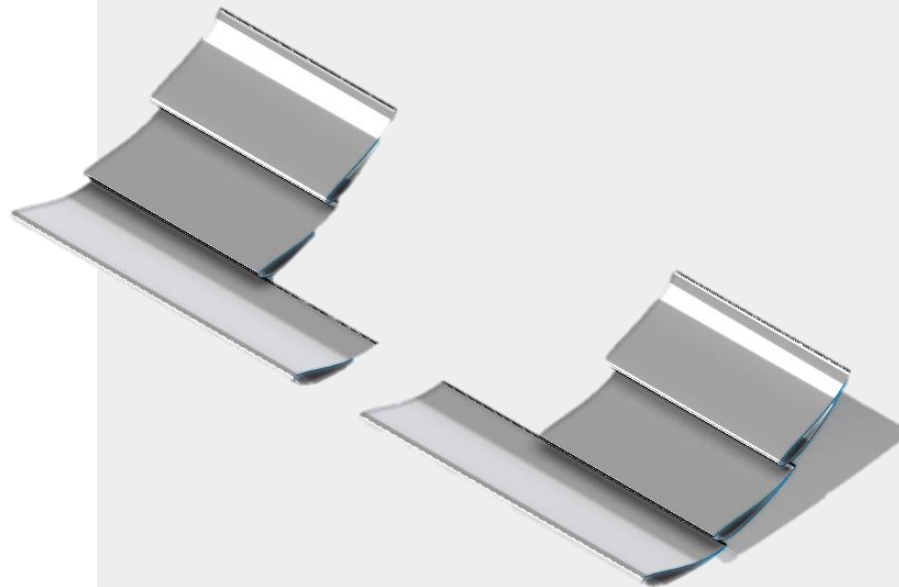
1. *Stiffness model*
2. *Linear static analysis*





Preprocessing

Importing geometry and
cleaning things up



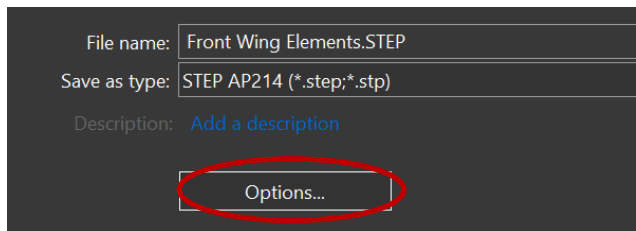
Front Wing Surface Bodies



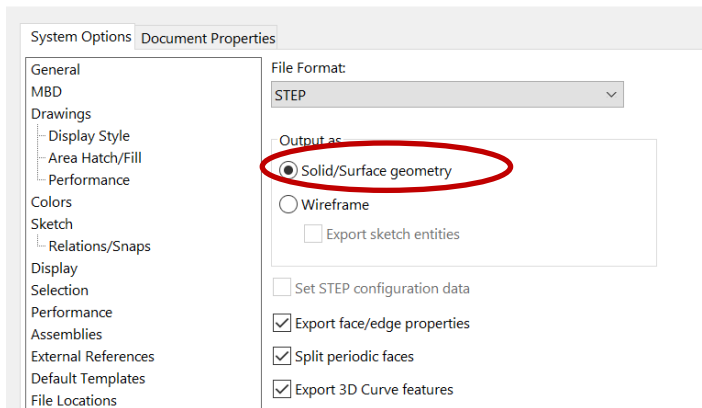
SolidWorks Cleanup



- 1) Open SolidWorks
- 2) Open the file  Front Wing Elements.SLDPRT from intro project folder
- 3) Delete the surface bodies that are shaded red
- 4) Export as step AP214 file
 - 1) Hit options and ensure that “Solid/Surface Geometry” is selected



System Options - STEP





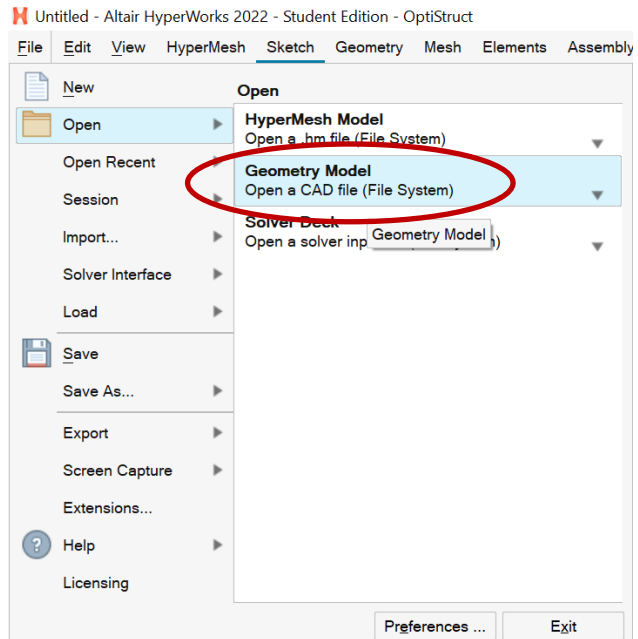
HyperWorks Import

Open up HyperWorks:



Import the earlier file by clicking:

- 1) File
- 2) Open
- 3) Geometry Model

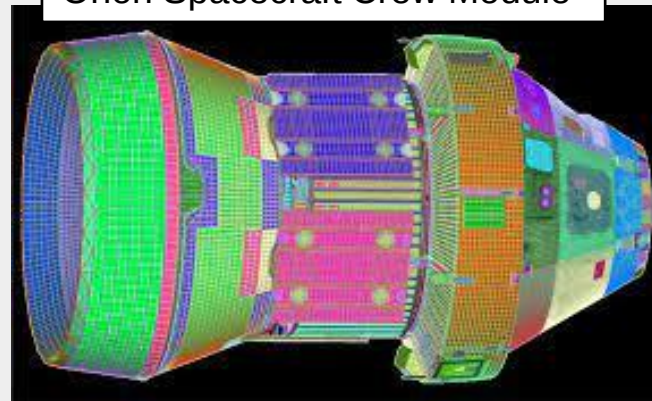




Discretization

How should you mesh the part?

Orion Spacecraft Crew Module



Space Shuttle Main Engine

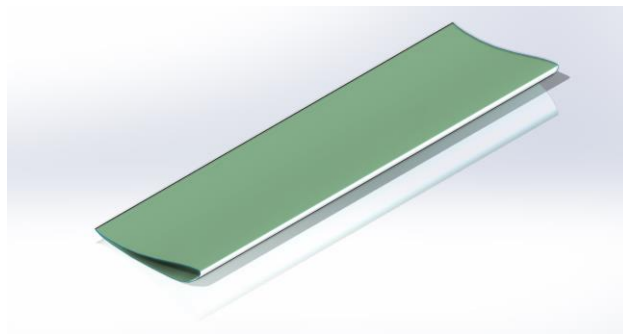


Discretization

Could we use 3d elements rather than 2d? What are the pros/cons of each?

For this example, we will be meshing the wing element as 2d quad elements.

Given the simplicity of the shape, we will easily be able to mesh the geometry as 100% quads which is ideal!



Idealized Part



Meshed Part

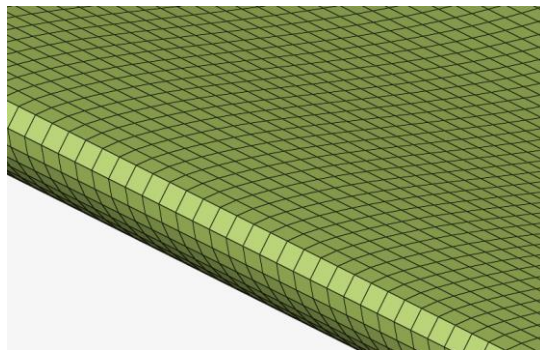


Discretization - How to mesh

In the mesh tab, select ***Generate 2D Mesh***



Select all surfaces, set a target element size of .125, and mesh





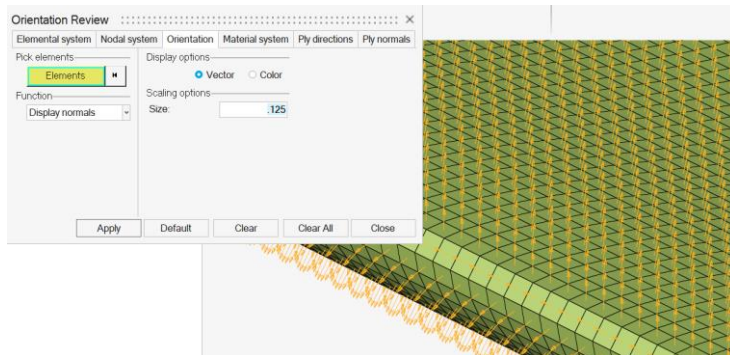
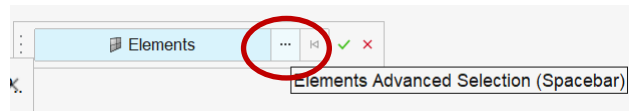
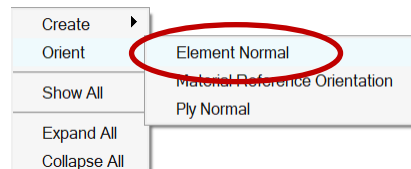
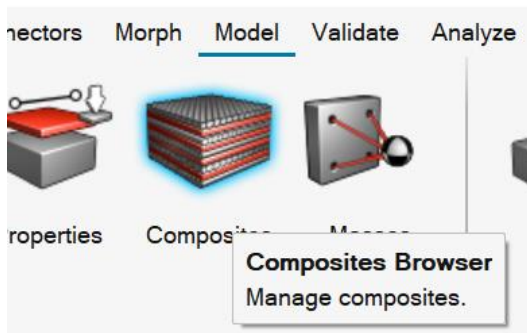
Composites Prep

- 1) Navigate to the Composites Browser
- 2) Right click in the browser
- 3) Select Element Normal
- 4) Click on the normals tab
 - 1) Expand the screen if you can't see all options

Click Elements, the "...", and then your wing components

Change Display Options to vector, and size to .125, and ensure that all normals are facing outwards

If normals are facing inwards, change Function to Reverse Normals and select Apply



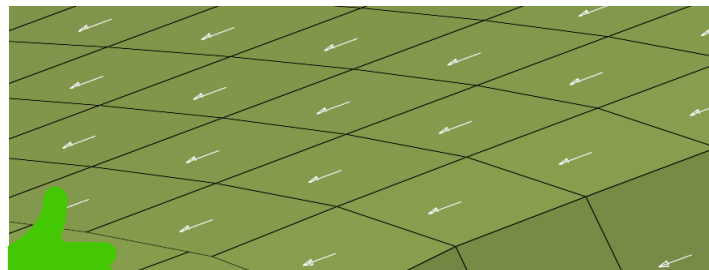
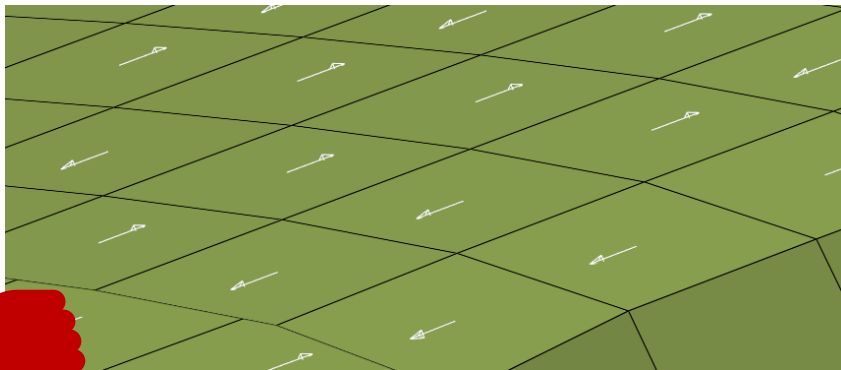
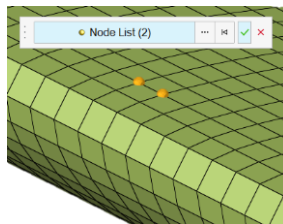
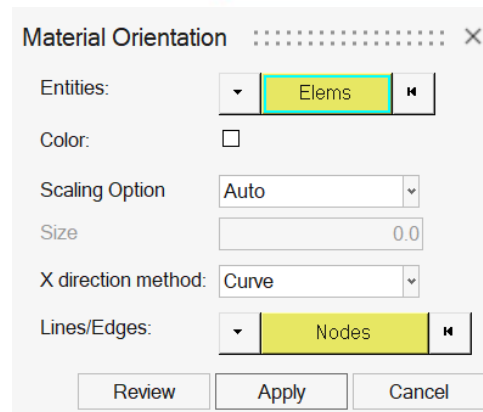
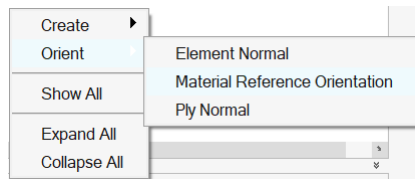


Composites Prep

Now we need to check element orientation.

All element orientations should be *in the same direction*, or your ply orientations will be meaningless.

Correct it by changing Lines/Edges to Nodes and selecting two nodes that form a vector in the spanwise direction.





Materials & Properties

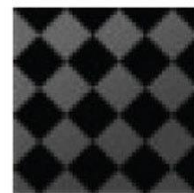
What are the properties of our parts?



UD



3K



12k

Example weaves of CF fabrics



Materials

Why are we using MAT8? Would a MAT1 be appropriate for our CF part?

Let's create a **MAT8 card** for your carbon fiber!

Name it “WR-4HS” and *change the card image from MAT1 to MAT8*.

Give your material the following values:

$E1 = E2 = 7660000$

$\text{Nu}12 = 0.042$

$G12 = 46400$

$\text{Rho} = 0.0001071$

What do you think these parameters mean?

tors (2)

s (2)

91)

(16080)

Create

Show All

Reverse

R

Load Step Inputs

Material

Multibody

Output Block

Modal Results Output

Parameter

Part

Plot

Name:	WR-4HS
ID:	1
Color:	
Include:	[Main Model]
Defined:	☒
Card Image:	MAT8
User Comments:	Hide In Menu/Export
E1:	7660000.0
E2:	7660000.0
NU12:	0.042
G12:	46400.0
G1Z:	
G2Z:	
RHO:	0.0001071



Properties - The PCOMPP Mindset

The PCOMPP property was designed to represent the intuitive physical process of laminating a carbon fiber part.

- 1) Create your shapes. You will do this by selecting a group of elements in the shape that you want. Think of this like creating a cardboard monocoque template for cutting plies.
- 2) Create unstacked plies. These plies must have a material, thickness, and orientation. Imagine you are now using your cardboard template to cut out a piece of carbon in a certain orientation. This essentially creates a big pile of plies that are cut and ready to be laid up.
- 3) Create a laminate. This is like taking a cut ply from your pile and laying it up. The order here matters!

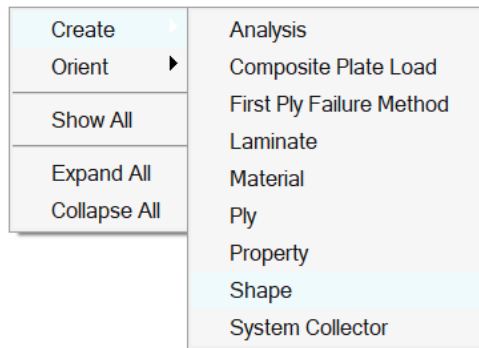




Properties - Ply Shapes

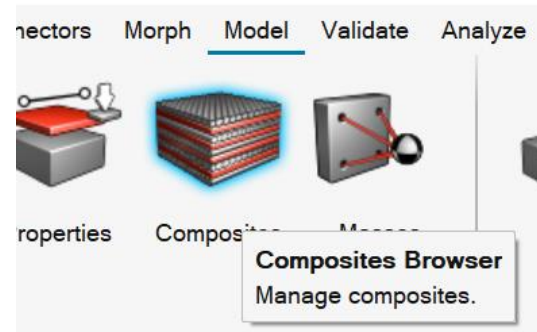
Now, let's create a PCOMPP property for your laminate. First, we must open the composites browser.

To get started, we will create a singular shape with every element in it.



Solver Keyword:	SET
Name:	full element
ID:	1
Include:	[Main Model]
Defined:	☒
Card Image:	SET_ELEM
Set Type:	non-ordered
Entity IDs:	 17642 Elements
User Comments:	Hide In Menu/Export
TYPE:	ELEM
SUBTYPE:	LIST
No of rows:	500
Data: ID:	

Every element



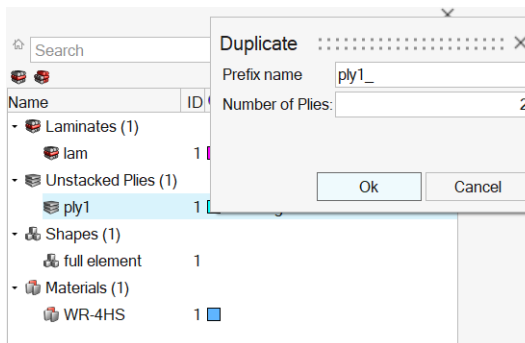
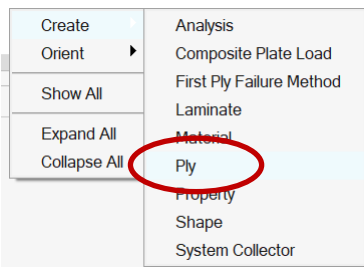


Properties - Unstacked Plies

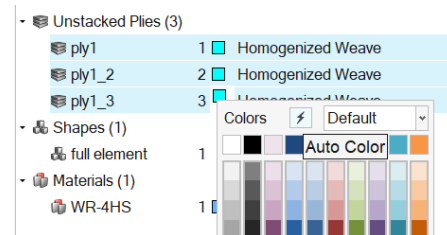
It's time to “cut” your plies. Imagine you are placing your template down (ply shapes) and cutting your plies at a certain angle.

Let's create our first ply like on the image on the right and then create 2 more duplicates.

You will now have a pile of unstacked plies sitting there, ready to be laid up. Recolor them too so it looks better.



Name	Value
Solver Keyword:	PLY
Name:	ply1
ID:	1
Color:	
Include:	[Main Model]
Card Image:	PLY
Thickness:	0.0095
Orientation:	0.0
Result request:	☒
Material:	(1) WR-4HS
Drape:	<Unspecified>
Shape:	1 Sets
Ply system:	<Unspecified>
Ply type:	Homogenized Weave



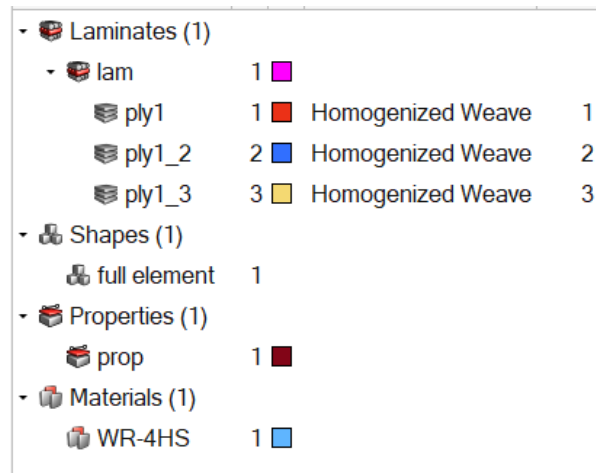
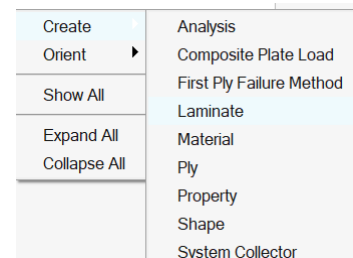


Properties - Create a Laminate

It's time to take your unstacked plies and “lay them up”.

Right click and create a laminate.

Select your plies and drag them to the laminate that you just created.



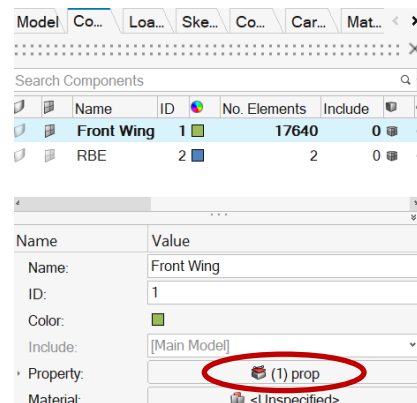
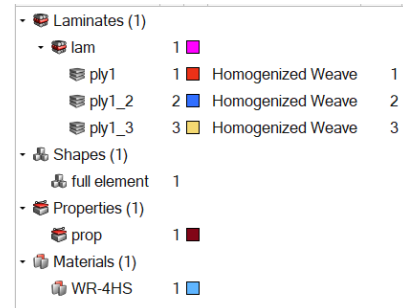
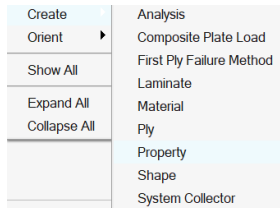


Properties - Create a Property

It's time to create a property that will apply your laminate characteristics to the mesh. While this may not make intuitive sense in the context of the ply-laminate workflow, it is the result of new developments being applied to an older, proven solver.

Right click and make a new property. Name it whatever and give it whatever color you want.

Go to the component tab and select your front wing component with the elements and set the property to the property that you just created



*If you can't find the component tab, you can access it by double-clicking components from the model tab

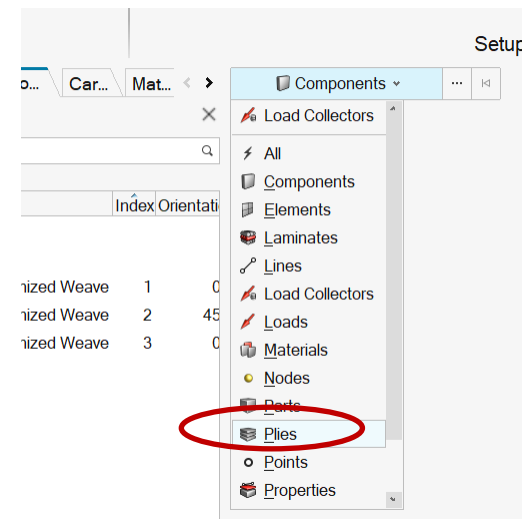
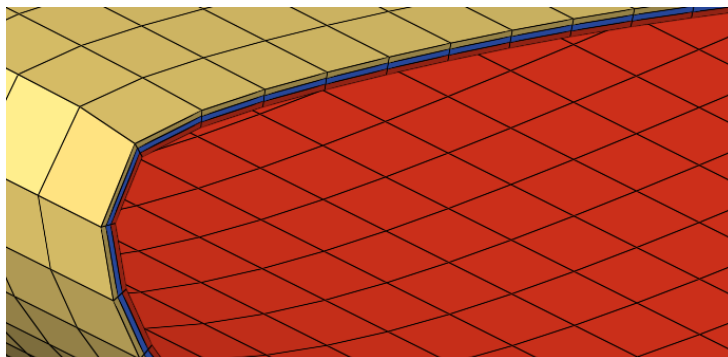


Properties - Visualizing

Now let's take a look at the laminate property that we have applied to our part!

Change the viewer to Plies and select the two visualization options at the bottom as such. Now, notice that your generic 2d shell elements are now being displayed as your laminate, with colors to differentiate between the plies and the layers have a thickness.

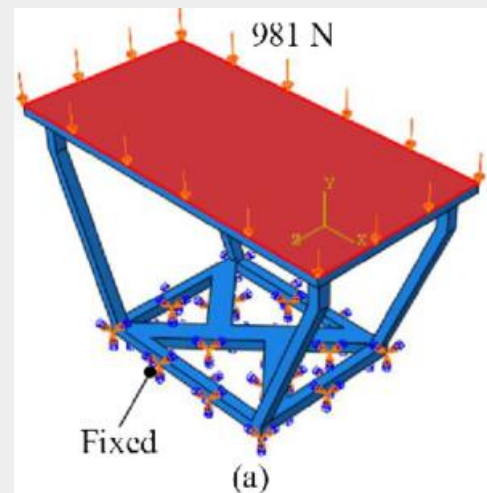
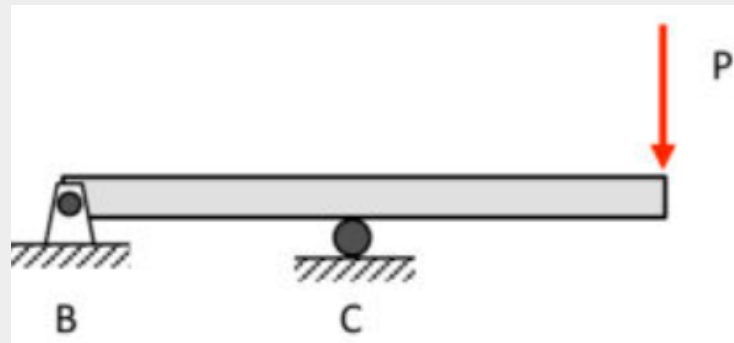
Great work!





Boundary Conditions

How should you model the part's environment?





Boundary Conditions: Loads

1) Open the project folder and run



WR-PressureMap-Cleaner.exe

1) When prompted for a file, select



Intro Project Pressure Map.csv

This is an automated script that cleans up the raw data out of StarCCM that is universally used by Wisconsin Racing for static aeroelastic simulations.

	A	B	C	D
1	Pressure (X (m)	Y (m)	Z (m)	
2	451.2711	-0.74992	-0.06422	0.040763
3	453.7723	-0.75149	-0.0635	0.040663
4	449.6631	-0.74992	-0.0635	0.040763
5	455.0158	-0.75149	-0.06405	0.040662
6	457.4751	-0.75306	-0.0635	0.040573
7	444.689	-0.74836	-0.0635	0.040883
8	445.5841	-0.74835	-0.06428	0.040883
9	458.5169	-0.75306	-0.06628	0.040572
10	454.5444	-0.75147	-0.06631	0.040663
11	461.2227	-0.75463	-0.06625	0.040495
12	460.5337	-0.75453	-0.0635	0.0405
13	450.0491	-0.7499	-0.06632	0.040764

Raw Data

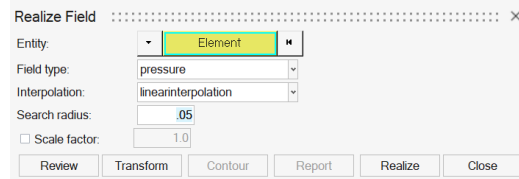
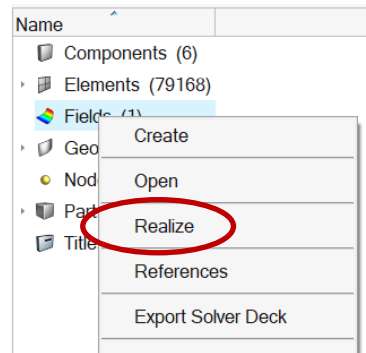
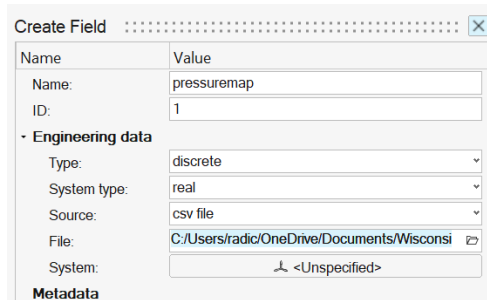
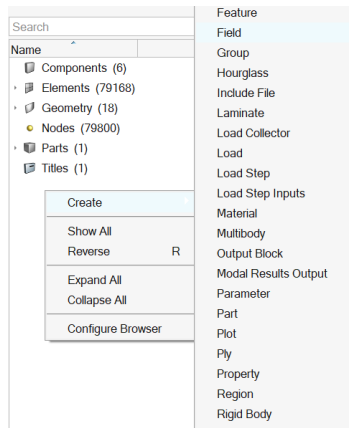
	A	B	C	D
1	X (in)	Y (in)	Z (in)	Pressure (psi)
2	-29.5244	-2.52843	1.604838	0.065451
3	-29.5862	-2.49999	1.600883	0.065814
4	-29.5245	-2.49999	1.604853	0.065218
5	-29.5862	-2.52169	1.60087	0.065994
6	-29.648	-2.49999	1.597378	0.066351
7	-29.4628	-2.49999	1.609558	0.064497
8	-29.4626	-2.53066	1.609553	0.064626
9	-29.6479	-2.60932	1.597301	0.066502
10	-29.5855	-2.6108	1.600903	0.065926
11	-29.7099	-2.60832	1.594302	0.066895
12	-29.706	-2.49999	1.594502	0.066795
13	-29.5237	-2.61084	1.604866	0.065274
14	-29.4624	-2.61185	1.609545	0.064562
15	-29.4013	-2.49999	1.614654	0.063683

Processed Data



Boundary Conditions: Loads

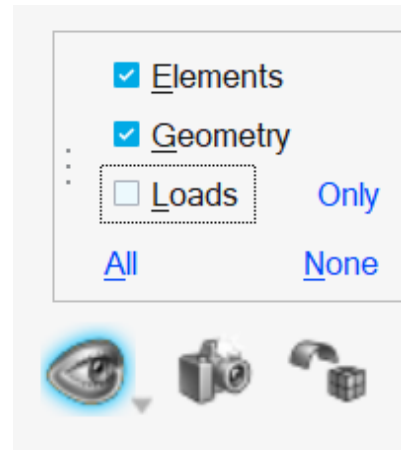
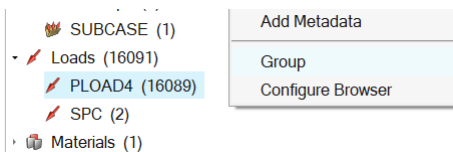
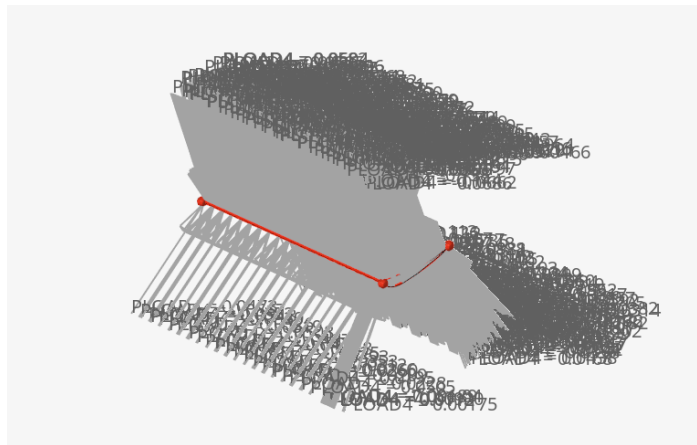
- 1) Right click, Create, Field
- 2) Set the “Create Field” parameters as shown
 - 1) Discrete, Real, CSV File
 - 1) Select the processed file that you created in the previous slide
- 3) Right click the Field that you created and hit Realize
 - 1) Select all elements, field type pressure
 - 2) Linear interpolate roughly half of your element size





Boundary Conditions: Loads

Group loads and/or hide them to make the sim easier to work on!





Boundary Conditions: SPC's

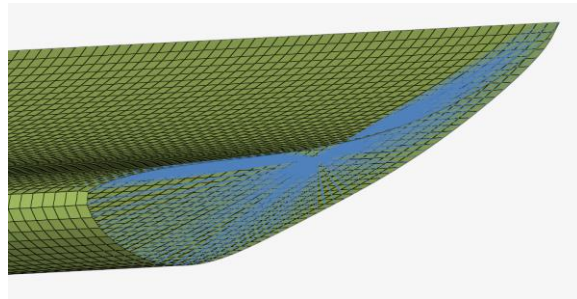
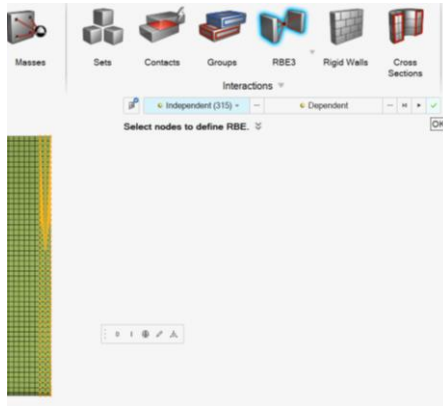
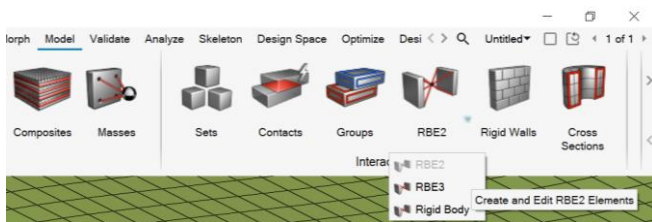
RBE2 elements add stiffness to your model. Is that okay here?

- 1) First, create a component called "RBE" to organize the RBE2 elements you are about to create



- 2) From a top view, drag to select 2-4 rows of nodes from the edge is independent and then hit the check mark

- 1) Constrain no more than 3/8" of nodes in from the edge





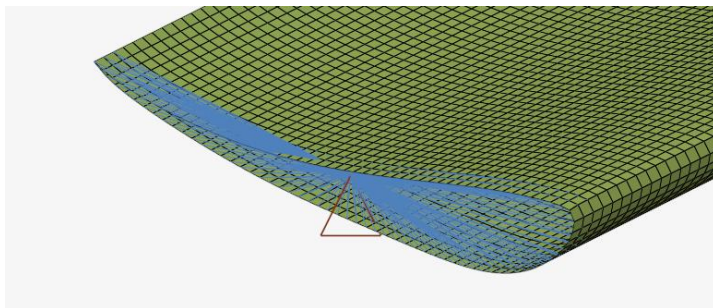
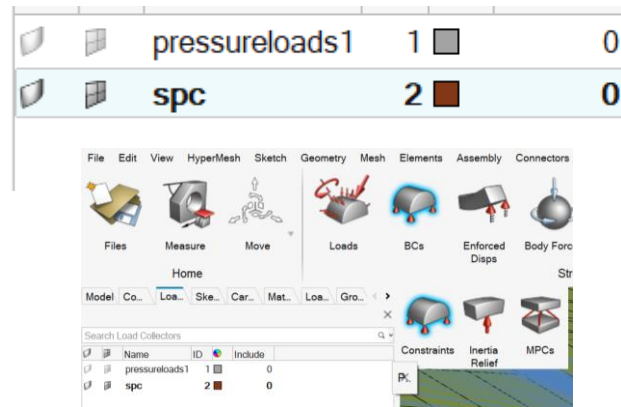
Boundary Conditions: SPC's

Create a load collector and *name it "spc"*

Constrain dof1, dof2, dof3, and dof5

Notice that we constrain all 6 DOF of the rigid body total!

Why also constrain dof5? Shouldn't dofs1-3 for each suffice?



☒ dof1	=	0.000
☒ dof2	=	0.000
☒ dof3	=	0.000
☐ dof4	=	0.000
☒ dof5	=	0.000
☐ dof6	=	0.000
load types =	SPC	



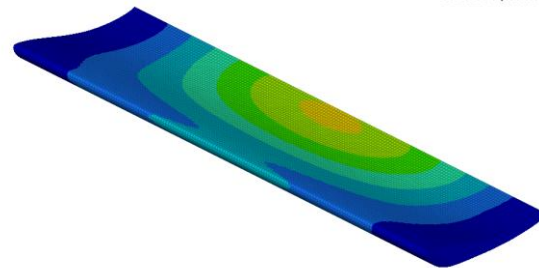
Running The Analysis

How do we send our information to the solver?

Contour Plot
Displacement(Mag)
Analysis system

7.288E-02
6.478E-02
5.669E-02
4.859E-02
4.049E-02
3.239E-02
2.429E-02
1.620E-02
8.098E-03
0.000E+00
No Result

Max = 7.288E-02
Grids 14121
Min = 0.000E+00
Grid 17747
Y
X



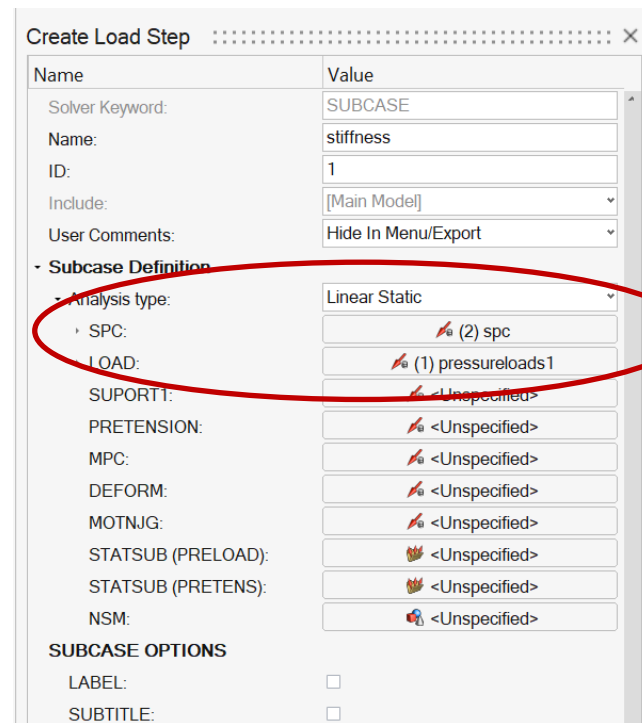
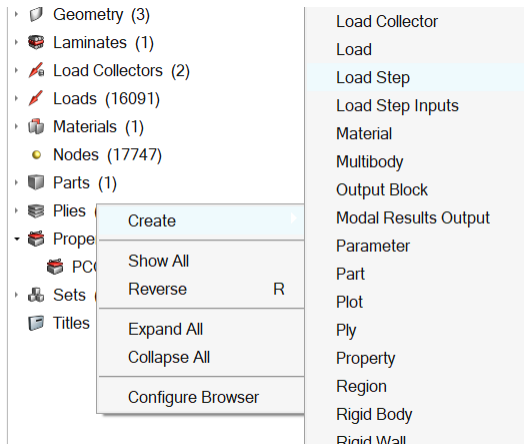
Default Wing Simulation Results



Running The Analysis: Load step creation

1) Create a Load Step

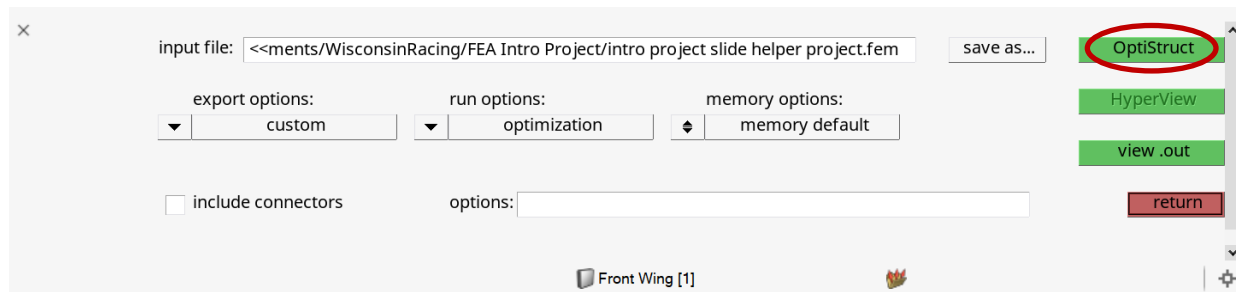
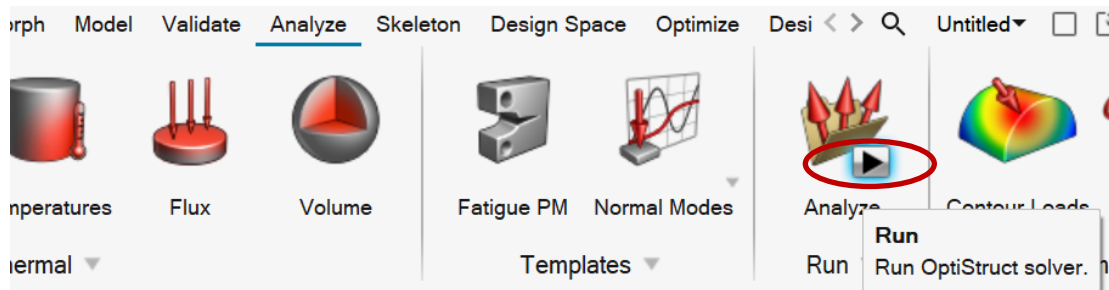
- 1) Set Analysis type to Linear Static
- 2) Set SPC to your SPC load collector
- 3) Set load to your pressure load collector





Running The Analysis: Sending to the Solver

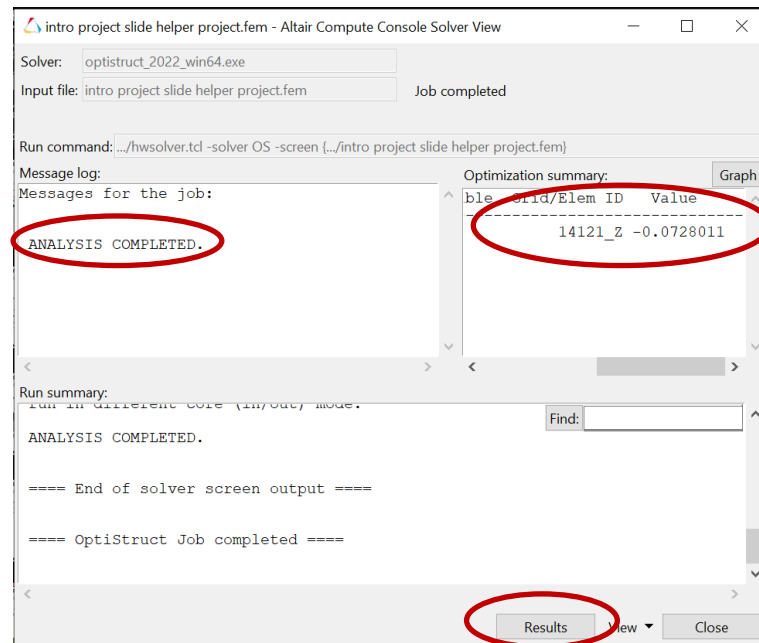
- 1) In the Analyze tab, click the play button
- 2) Click “Optistruct” at the bottom





Running The Analysis: Viewing Results

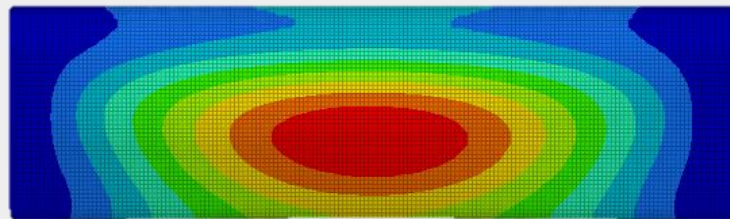
- 1) When the analysis is complete, you will see “ANALYSIS COMPLETED.” in the Message log
- 2) Next, check the Optimization summary box to see your displacement. Is it a reasonable value?
- 3) Click results to view the results!





Post Processing

Viewing/Understanding Results



Example wing deformation plot

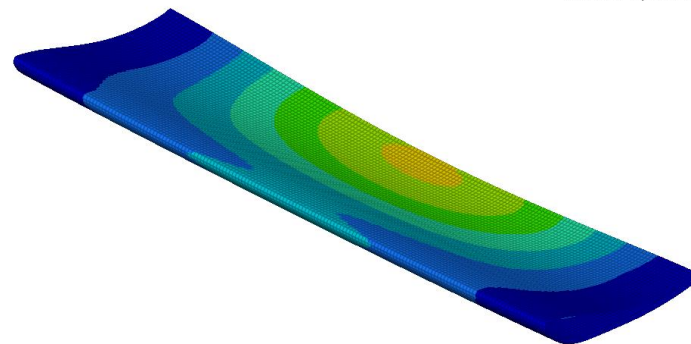
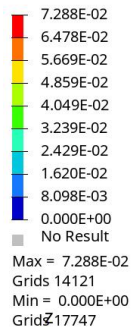


Post Processing:

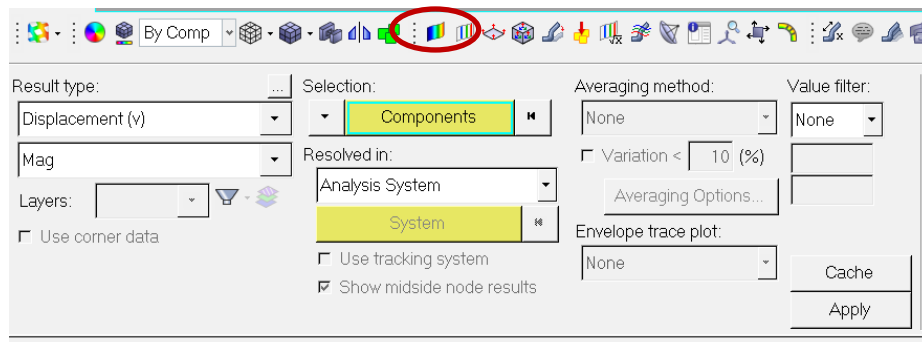
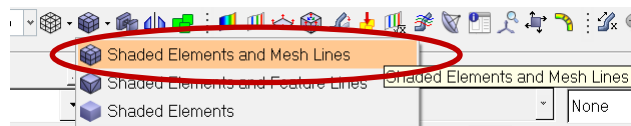
Select Shaded Elements and Mesh Lines

Click the contour icon and then apply

Contour Plot
Displacement(Mag)
Analysis system



1: 1
Subcase 1 (stiffness) : Static Analysis : Frame 4





Challenge Introduction

Wing Structure Optimization



WR223 Front Wing



Wing Optimization Challenge

Your objective is to ***minimize the mass of the wing element while keeping the maximum deflection under .025 inches***, given the following set of constraints.

- 1) The design must be manufacturable.
 - 1) If you optimize beyond something that can be done with hand-layup, you must provide a short explanation on the techniques used.
 - 2) You may not use unidirectional fabric as either of your outermost layers.
- 2) You may not edit the provided material cards or loads in any way.
- 3) Your simulation must be a good faith approach in solving the problem. The aim should not be to cheat the system, but rather apply your knowledge to make a winning design.

Here are some ideas to get you started:

- Play around with different materials/layups
 - Try some unusual layups, they can be unbalanced or nonsymmetric
- Experiment with adding 1 or more spars
- Experiment with adding 1 or more ribs

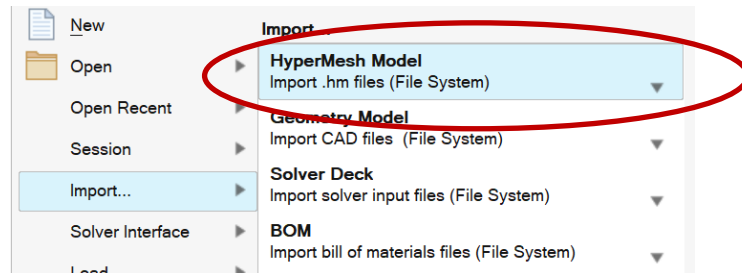


Wing Optimization Challenge - Adding Materials

In order to make the challenge more interesting, we have made a few other materials available to use. Let's add them in now!

- 1) Click File > Import > HyperMesh Model
- 2) Select the file  Intro Project Challenge Materials.hm from the challenge resources folder
- 3) Advance through the dialogue, all defaults are acceptable

They have already been made into Unstacked Plies so duplicate them, change orientations, and change shapes as desired!



Name	ID	Product	Index	Orientation	Thickness
- Unstacked Plies (4)					
WR-4HS	1	Homogenized Weave		0.0	0.0095
UNI	2	Unidirectional		0.0	0.0065
SPREAD-TOW	3	Homogenized Weave		0.0	0.0065
TWILL	4	Homogenized Weave		0.0	0.0095
- Materials (4)					
WR-4HS	1				
UNI	2				
SPREAD-TOW	3				
TWILL	4				

Materials to choose from



Wing Optimization Challenge - Deliverables

In order to verify the integrity of your sim,
we will analyze output files with a script

- 1) Open your SUBCASE load step
- 2) Scroll down in SUBCASE OPTIONS until you see “SPCF” and click the check mark
- 3) Set the format to OPTI
- 4) Repeat steps with DISPLACEMENT

- Load Collectors (2)
- Load Steps (1)
- SUBCASE (1)**
- Loads (3)

Solver Keyword:	SUBCASE
Name:	stiffness
ID:	1
Include:	[Main Model]
User Comments:	Hide In Menu/Export
Subcase Definition	
SUBCASE OPTIONS	
LABEL:	☐
SUBTITLE:	☐
ANALYSIS:	☒
TYPE:	STATICS
ASSIGN:	☐

DISPLACEMENT:	☒
DISPLACEMENTS_NUM =:	1
1	
SORTING:	
FORMAT:	OPTI
FORM:	

SPCF:	☒
SPCFORCES_NUM =:	1
1	
SORTING:	
FORMAT:	OPTI
SPCF_FORM:	



Challenge Submission Checklist

Run through these criteria before your submission:

- ☐ My design deflects less than .025"
- ☐ My design is manufacturable by some technology, which I can name
- ☐ I don't have unidirectional fabric on either of the outermost surfaces
- ☐ I did not modify the material cards in any way
 - ☐ Single ply thicknesses must remain at their default values.
- ☐ I did not modify the loads in any way
- ☐ I have my .spcf, .disp, and .fem files from the previous slide ready to submit

Sweet, now submit your design using the .bat file in the folder. We can't wait to see what you come up with!