Failure of Notched Laminates Under Out-of-Plane Bending Phase VII

Fall 2014 Meeting

Mitchell Daniels, Levi Suryan,

& John P. Parmigiani,
Oregon State University

Failure of Notched Laminates Under Out-of-Plane Bending. Phase VII

- Motivation and Key Issues

  Develop analysis techniques useful in design of composite aircraft structures under out-of-plane loading (bending and shear)

- Objective

  Determine failure modes and evaluate capabilities of current models to predict failure

- Approach
  - Experiments: Out-of-plane shear (mode 3 fracture)
  - Modeling: Progressive damage development and delamination (Abaqus) under Mode 3 fracture

Failure of Notched Laminates Under Out-of-Plane Bending. Phase VII

- Principal Investigators & Researchers
  - John Parmigiani (PI); OSU faculty
  - M. Daniels, L. Suryan; OSU grad students
- FAA Technical Monitor
  - Curt Davies
  - Lynn Pham
- Other FAA Personnel Involved
  - Larry Ilcewicz
- Industry Participation
  - Gerry Mabson, Boeing (technical advisor)
  - Tom Walker, NSE Composites (technical advisor)

Project Overview

Phase I (2007-08)

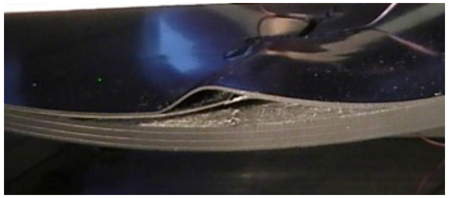
- Out-of-plane bending experiments w/composite plates
- Abaqus modeling with progressive damage

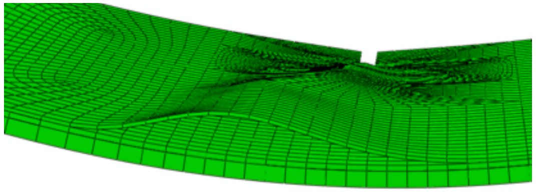
Phase II (2008-09)

- Abaqus modeling with buckling delamination added
- Sensitivity study of (generic) material property values

Phase III (2009-10)

- Abaqus modeling w/ more delamination interfaces





Project Overview

Phase IV (2010-11)

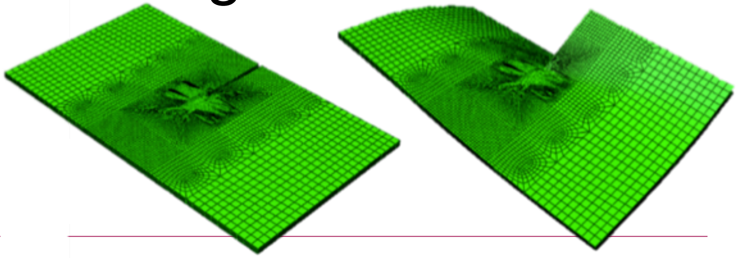
- Begin out-of-plane shear experiments
- Continue out-of-plane bending modeling

Phase V (2011-12)

- Complete out-of-plane shear experiments
- Begin out-of-plane shear modeling

Phase VI (2012-13)

- Continue out-of-plane shear modeling
  - Abaqus Standard
  - Abaqus Explicit
  - Helius MCT



Project Overview

Phase VII (2013-14)

- Comprehensive report on Phase VI work for Boeing
- Evaluation of solid vs. shell elements in modeling
- Improvement to Abaqus Explicit models
- Explore damage softening parameters in SCA (Simulation Composites Analysis, formerly Helius: MCT)
- Sensitivity study of material properties to explore possibility of inaccuracies in material properties

A Center of Excellence AMTAS Advanced Materials in Transport Aircraft Structures

Today's Topics

- Brief review of computational Model
- Evaluation of Solid Versus Shell Elements
- Improvements to Abaqus Explicit
- Exploration of SCA damage parameters
- Material Parameters Sensitivity Study
- Summary of Mode III Analysis
- Conclusions

Today's Topics

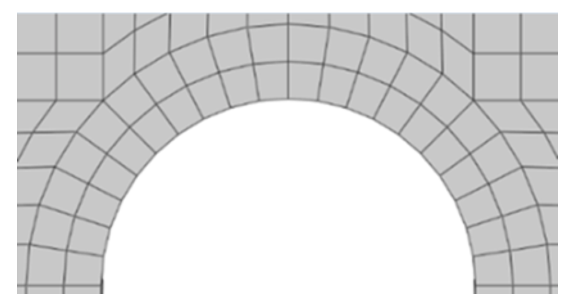
- Brief review of Computational Model
- Evaluation of Solid Versus Shell Elements
- Improvements to Abaqus Explicit
- Material Parameters Sensitivity Study
- Future Work
- Summary of Mode III Analysis
- Conclusions

Computational Model

- Uses Hashin failure criteria
- Quasi-static analysis and non-linear geometry turned on
- Panel: Continuum shell, reduced integration elements (SC8R)
- Mesh defined around notch tip
- Grips: Continuum, 3-D, 8 node, reduced integration element (C3D8R)
- Boundary conditions implemented by grips

Grip allowed to displace in z-axis and allowed to rotate about x-axis

Grip is fixed, but allowed to rotate about x-axis

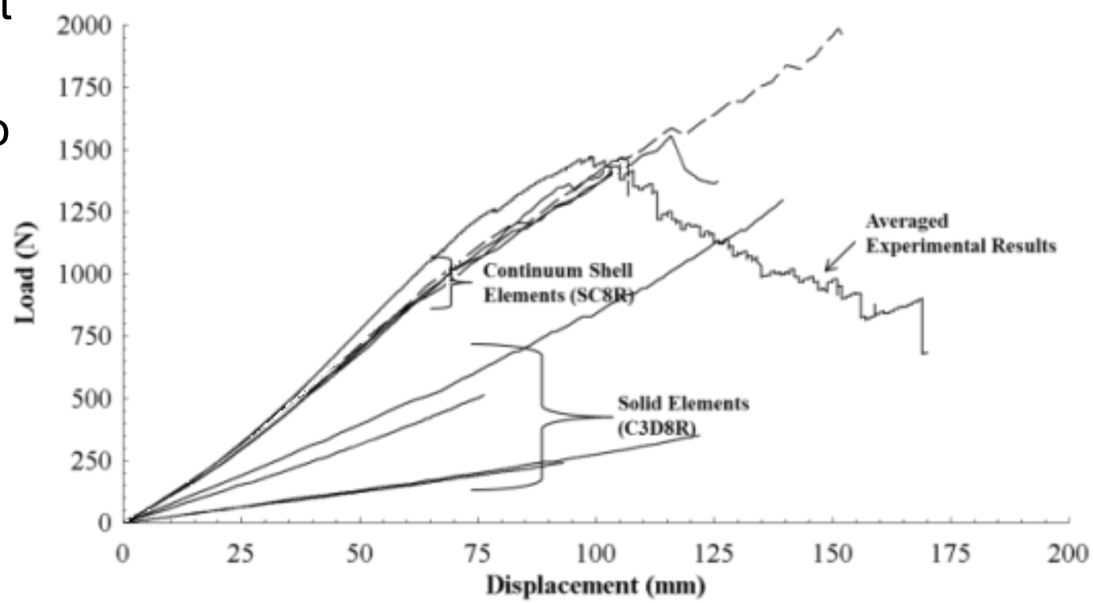


Today's Topics

- Brief review of Computational Model
- Evaluation of Solid Versus Shell Elements
- Improvements to Abaqus Explicit
- Exploration of SCA damage parameters
- Material Parameters Sensitivity Study
- Summary of Mode III Analysis
- Conclusions

Solid Vs. Shell Elements

- Task: Explore the effect of using solid elements versus shell elements in Models
- Approach: Compared elements with Simulation Composites Analysis (SCA) (formerly Helius:MCT) because solid elements not available using Hashin damage criteria with Abaqus Standard or Explicit
- Results: Solid elements are not recommended
  - Fine mesh requirements to simulate laminates
  - Becomes computationally prohibitive before solution convergence
  - Limits accuracy of simulations

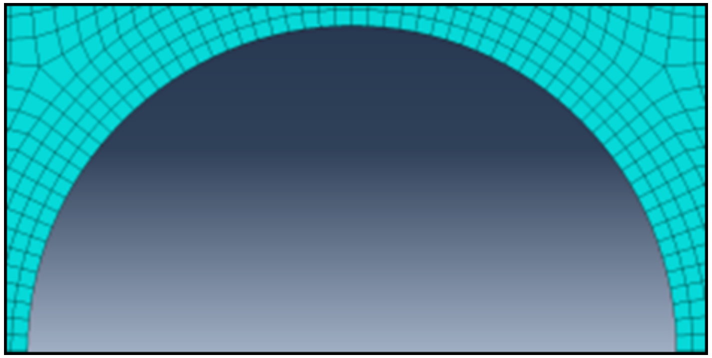


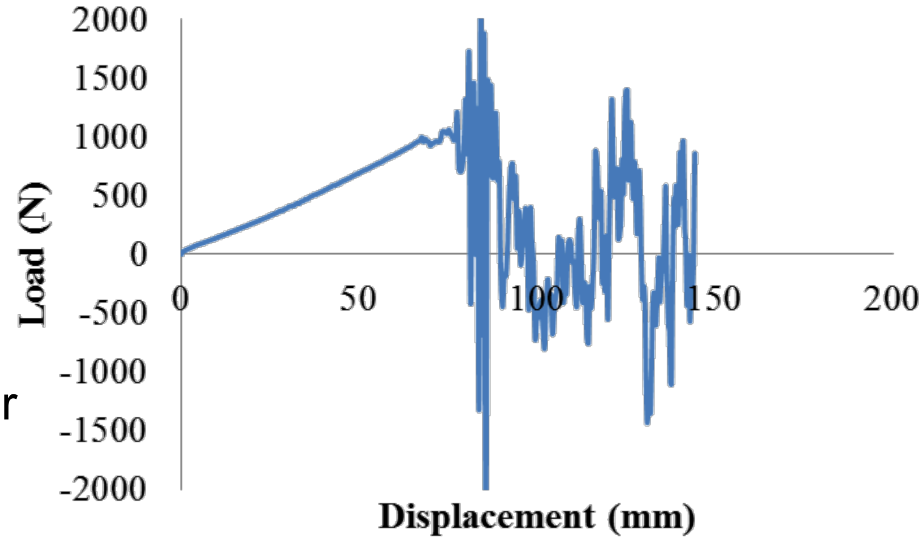
Today's Topics

- Brief review of Computational Model
- Evaluation of Solid Versus Shell Elements
- Improvements to Abaqus Explicit
- Exploration of SCA damage parameters
- Material Parameters Sensitivity Study
- Summary of Mode III Analysis
- Conclusions

Improvements for Abaqus Explicit-Increased Mesh Density

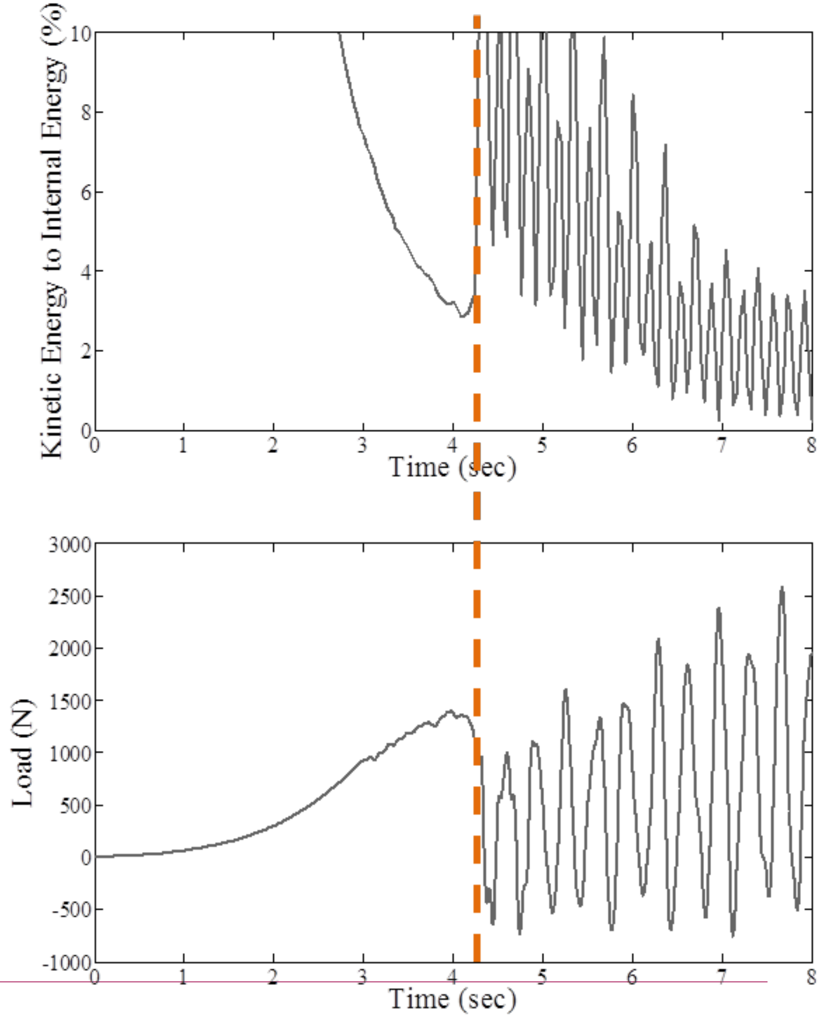
- Task: Reduce or eliminate noise from explicit solver by increasing mesh density
- Approach: Increased mesh density from 20 elements around the notch tip to 32 and 64 elements around notch tip
- Results:
  - Increasing mesh density not effective for reducing noise
  - Noise can lead to inaccuracies and requires filtering
  - Max load results discussed later in presentation





Improvements for Abaqus Explicit- Mass Scaling

- Task: Reduce the time of Abaqus explicit solver analysis
- Approach: Mass Scaling
  - Conditionally stable solver yields long run times due to small time steps
  - Mass scaling specifies minimum time step
  - Scales mass to reach min time step
  - Value: 5x10-6 s
- Results: Drastically reduces solver time from ~230 hours to ~30 hours
- Quasi static condition considered valid: oscillations occur after maximum load
- Max load comparison on next slide



Improvements for Abaqus Explicit- Max Load Results

- Compared mass scaling and increasing mesh density's effect on max load
- 40 ply layups or 64 element max load not considered due to computation time restraints and large file sizes making extracting max loads impractical
- Increased mesh density does not produce more accurate results
  - Hashin Damage mesh dependent in FEA solvers
  - Element characteristic length (L^c) decreases strain to failure (δ_f) by $\delta_f = 2G_{l,c}/XL^c$
- Mass scaling with 20 elements show best agreement with experiments:
  - Mass scaling parameter of simulation, limits ability as a predictive tool
  - Mass scaling strongly suggested to reduce computation times

Maximum Load Percent Deviation from Experiments:

Model	10 % 0°-20 Plies	30 % 0°-20 Plies	50 % 0°-20 Plies
20 elements-Mass Scaling	10.9%	-1%	-19%
20 elements-No Mass Scaling	21.70%	23.80%	56%
32 elements-Mass Scaling	1.30%	-15.80%	-27.90%

Today's Topics

- Brief review of Computational Model
- Evaluation of Solid Versus Shell Elements
- Improvements to Abaqus Explicit
- Exploration of SCA damage parameters
- Material Parameters Sensitivity Study
- Summary of Mode III Analysis
- Conclusions

Abaqus with SCA- Overview

- Autodesk Simulation Composite Analysis (SCA): Plug-in that applies damage criteria to fiber and matrix, formerly Helius:MCT

Volume fractions

$$\sigma^c = \phi_f \sigma^f + \phi_m \sigma^m$$

Average stress of composite, fiber, and matrix respectively

- <u>Task</u>: Improve selection of instant stiffness degradation parameters for composite panels in Mode III shear
- <u>Approach</u>: Evaluate FEA results for convergence, maximum load, and damage trajectories
- <u>Results</u>: following slides

Abaqus with SCA- Instant Stiffness Degradation Parameters

- Instant degradation parameters D_m and D_f reduce matrix and fiber stiffnesses when damage criteria are met
- Parameters are user defined
- 8 combinations of D_m and D_f were evaluated, starting with those recommended by Autodesk
  - Shotgun approach for selecting combinations taught value of designed experiments
- Combinations effected convergence and load versus displacement results

Run	Instant Degradation Values		Notes
	Matrix (D_m)	Fiber (D_f)	
1	1.00E-01	1.00E-06	SCA default values
2	1.00E-01	1.00E-04	Large D_f
3	1.00E-02	1.00E-05	Small D_m, large D_f
4	1.00E-03	1.00E-06	Very small D_m
5	1.00E-02	1.00E-04	Small D_m, large D_f
6	1.00E-03	1.00E-03	Very small D_m, very large D_f
7	1.00E-01	1.00E-04	Large D_m, small D_f
8	2.00E-01	1.00E-06	Very large D_m, very small D_f

Abaqus with SCA- Convergence Results

- Convergence issues limited available softening parameter combinations (33 of 48 runs failed)
- No stiffness degradation parameter in study yielded convergence for all layups, meaning convergence is limiting accuracy of results

		Instant Stiffness Degradation Combination							
		1	2	3	4	5	6	7	8
Layup	40P-50%	F	F	F	F	S	S	F	F
	40P-30%	F	S	S	S	S	S	F	F
	40P-10%	F	F	S	S	F	S	F	F
	20P-50%	F	F	S	F	F	F	F	F
	20P-30%	F	F	F	S	S	S	F	F
	20P-10%	F	F	S	F	F	F	F	F
Success Rate		0.0	0.17	0.67	0.5	0.33	0.67	0.0	0.0

S: Maximum load reached (success)

F: No max load before model divergence (failure)

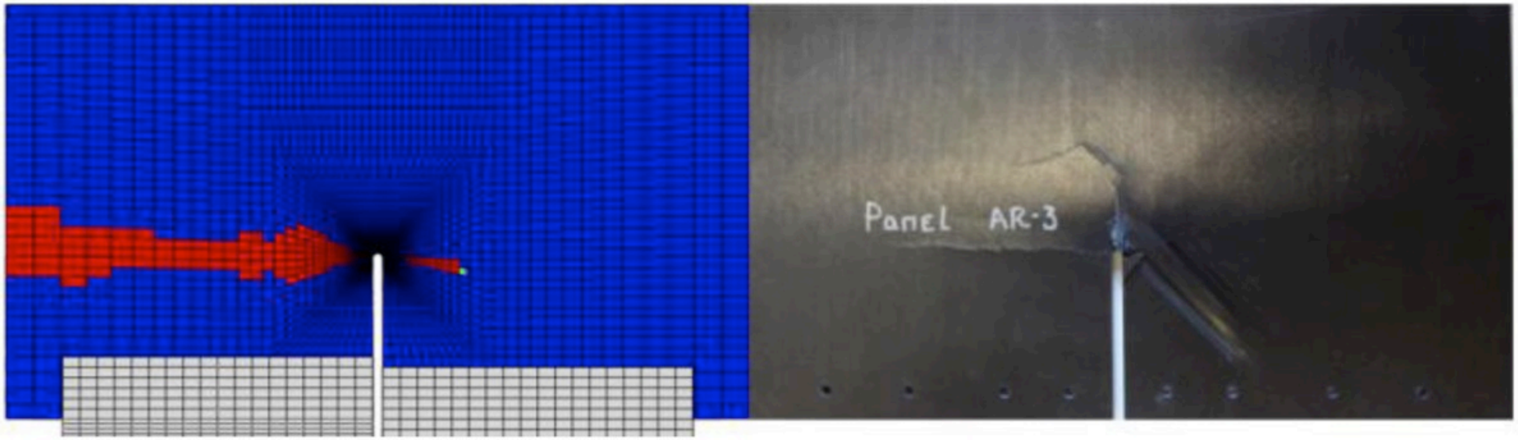
Abaqus with SCA- Maximum Load Results

- Maximum load of FEA model for each stacking sequence was compared to average maximum load of corresponding experimental group
- Superior stiffness degradation parameter combination selected by maximum load accuracy

Layup	ISD Combo	Experimental Max Load (lb)	FEA Max Load (lb)	Percent Deviation
40P-50%	6	1230	1115	9.8%
40P-30%	5	1283	1238	3.6%
40P-10%	4	921.9	1115	18.9%
20P-50%	3	403.6	296.5	30.6%
20P-30%	6	344.2	346.9	0.8%
20P-10%	3	282.9	299.8	5.8%
		Average Percent Deviation:		11.6%

Abaqus with SCA- Damage Trajectory Results

- Damage trajectories indicate quality of FE analysis technique
- Two of three recurring damage trajectories successfully modeled:
  - Splitting damage (successful), from notch tip in perpendicular direction
  - Diagonal damage (successful), from notch tip in direction of surface ply fibers
  - Self-similar damage (unsuccessful), from notch tip in the direction of the notch
- Self-similar damage involved extensive delamination; not modeled by single element through thickness



Today's Topics

- Brief review of Computational Model
- Evaluation of Solid Versus Shell Elements
- Improvements to Abaqus Explicit
- Exploration of SCA damage parameters
- Material Parameters Sensitivity Study
- Summary of Mode III Analysis
- Conclusions

Sensitivity Study-Tasks

- Determine which Hashin damage parameters (listed below in table) have significant effect on max load during finite element simulations
- Using results from study determine if possible errors in nominal parameter values could explain errors in simulations and which values of parameters could yield experimental results

Symbol	Definition
XT	tensile strength in 1-dir.
XC	compressive strength in 1-dir.
YT	tensile strength in 2-dir.
YC	compressive strength in 2-dir.
SL	longitudinal (in-plane) shear strength
SC	transverse shear strength
Gft	Energy requried to fully damage a ply using fiber tension only
Gfc	Energy requried to fully damage a ply using fiber compression only
Gmt	Energy requried to fully damage a ply using matrix tension only
Gmc	Energy requried to fully damage a ply using matrix compression only

Sensitivity Study- Approach

- 10 factor, 2 level partial factorial
  - 1/16th fractional factorial: 2^{10-6}
  - Levels: ±20% from nominal values
  - Resolution IV: No first order interactions confounded with 2nd-order interactions
- Only 20 ply panels considered due to the large number of runs
- Results from implicit and explicit solvers considered
- Used normal probability plots used to determine significance
- Secondary factorial experiment used if confounding factors significant
- Regression modeling used to determine which parameter values yield experimental results

Sensitivity Study- Significance Results

- XT and XC (longitudinal tension and compression strength) shown to be the most significant parameters for mode III loading case
- Other parameters showed significance for certain layups but have small effects when compared to XT and XC
- Solver types yields different results
  - Explicit and implicit solvers different techniques yielding slightly different results
  - Solver tools like mass scaling can change results

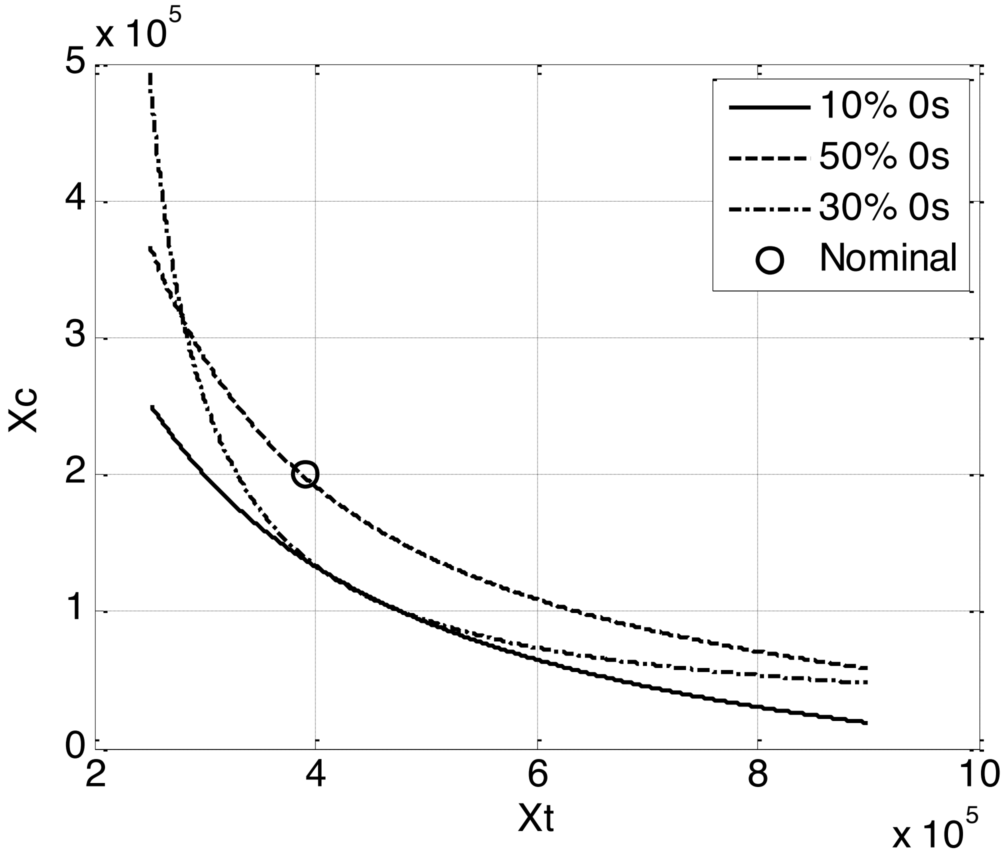
Layup	Analysis Method	Significant Parameters in Order of Significance
20P/10%	Abaqus Standard	XT, XC, SL
20P/30%	Abaqus Standard	XT, XC, XT-XC Interaction
20P/50%	Abaqus Standard	XT, XC
20P/10%	Abaqus Explicit	XT, XC, SL, XT-XC Interaction, GFC, GFT
20P/30%	Abaqus Explicit	XT, XC, XT-XC Interaction, SL
20P/50%	Abaqus Explicit	XT, XC

Sensitivity Study- Regression Approach

- XT and XC main factors considered for regression due to significance
- Fitted model to sensitivity study data considering all ten factors and all second order interaction
- Optimization in statistics program for experimental max load with all factors considered, could not reach agreement between layups
- Varied XT and XC
  - compared results to experimental average maximum load
  - No agreement between different layups
  - Ran first ply failure analysis to calculate the strength of laminates
  - Used new strengths in regression to check for agreement

Sensitivity Study- Regression Results

- Figure shows variation of XT and XC with the lines representing the constant maximum load according to experimental results
- Material parameter errors are not able to explain simulation errors
  - Different layups do not agree on specified values of XT and XC
  - Could not predict simulation results using nominal values
  - Only accurately predicts FEA response for parameter values near levels in study (±20% Nom)
  - Suggests nonlinearity and other factors in FEA not captured by regression model



Today's Topics

- Brief review of Computational Model
- Evaluation of Solid Versus Shell Elements
- Improvements to Abaqus Explicit
- Exploration of SCA damage parameters
- Material Parameters Sensitivity Study
- Summary of Mode III Analysis
- Conclusions

Summary of Mode III Analyses from Phase VI and VII

- Summary of all Mode III analyses from Phase VII (Explicit and SCA) and Phase VI (Standard)
- Hourglass stiffness scaling used in Standard to prevent large element deformations
- Explicit shows best agreement with experimental average maximum loads after mass scaling
- SCA and Standard had convergence difficulties
- Adding cohesive elements improved standard accuracy, but could not achieve convergence for explicit and SCA

	Standard				Explicit	Standard with SCA
Panel Layup	1 Element Thick- No HGSS	1 Element Thick- HGSS	2 Element Thick- HGSS	3 Element Thick- HGSS	1 Element Thick- Mass Scaling	1 Element Thick
20-10% 0°	31%	15%	20%	-1%	11%	6%
20-30% 0°	**DNC**	19%	21%	18%	-1%	**DNC**
20-50% 0°	-17%	-16%	-23%	**DNC**	-19%	-27%
40-10% 0°	40%	22%	18%	17%	18%	22%
40-30% 0°	**DNC**	3%	-7%	22%	-4%	-4%
40-50% 0°	-3%	-5%	-4%	**DNC**	-10%	**DNC**
Average	**23%**	**13%**	**16%**	**15%**	**11%**	**15%**

HGSS: Hourglass Stiffness Scaling. DNC: Did Not Converge.

Today's Topics

- Brief review of Computational Model
- Evaluation of Solid Versus Shell Elements
- Improvements to Abaqus Explicit
- Exploration of SCA damage parameters
- Material Parameters Sensitivity Study
- Summary of Mode III Analysis
- Conclusions

Conclusions from Phase VII

- Shell elements yield better results than solid elements
- Sensitivity study of material parameters showed longitudinal tension and compression strengths are most significant parameters
  - XT and XC shown to be most significant parameters for mode III
  - Regression models failed to predict desired property values
  - Material parameters considered can not predict discrepancy between experiment and simulation alone
- SCA is accurate and efficient, but has severe convergence issues
- Explicit yields best agreement with experiments
  - Refining mesh density does not achieve better agreement with experiments
  - Mass scaling drastically reduces computation time
  - Most recommended of current tools
  - Limited as predictive tool due to mass scaling being element of solver

Questions

Out-of-Plane Shear: Summary of Experimental Results

- Maximum applied load (failure load)

	Max Force per Test [kN]						
Layup (#plies / % zero degree)	1	2	3	4	5	6	**MEAN**
40/50%	5.552	5.345	5.122	6.103	5.395	5.321	**5.473**
40/30%	5.342	5.363	6.061	5.616	6.176	5.690	**5.708**
40/10%	3.891	4.161	4.112	4.016	4.277	4.148	**4.101**
20/50%	1.751	1.859	1.929	1.691	1.740	1.801	**1.795**
20/30%	1.484	1.541	1.541	1.456	1.527	1.638	**1.531**
20/10%	1.290	1.215	1.258	1.254	1.198	1.336	**1.259**

Why Continuum Shell Elements vs. Solid Elements

- Solid elements can be laminated but max order of variation of the displacement is quadratic
  - Hence strain variation is at most linear
  - Insufficient to model variation of strain through thickness of laminate
- Potential Solution: stack solid elements at one element per lamina
  - In-plane dimensions can not be > 10x thickness
  - Requires a really fine mesh
- Alternate Solution: Use continuum shell elements
  - Does not have the same problems as a solid element
  - Can have multiple plies through the thickness
  - Also can be stacked for using with grips and delamination
- Laminate stacking sequence was constructed using Composite Layup in Abaqus – define material prop' per ply

Scaling Hourglass Stiffness

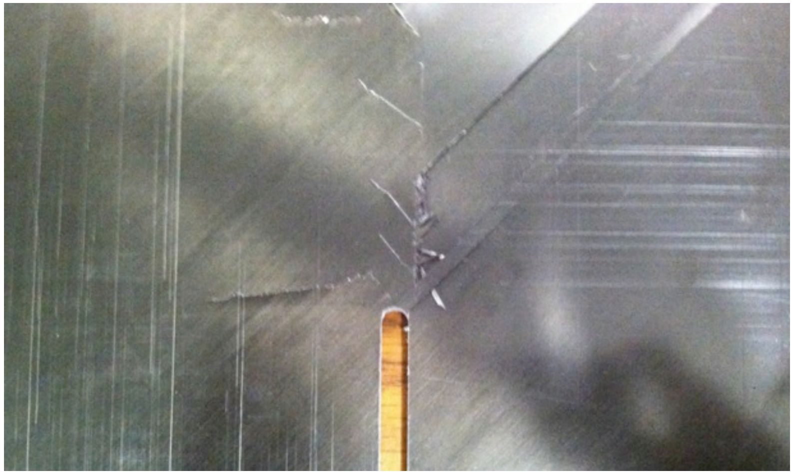
- Default hourglass stiffness was scaled to prevent severe element deformation
- Pure stiffness approach was recommended for quasi-static analysis
- Three user defined scaling factors

Factor	Description	Typical Range
	Hour glass stiffness scaling factor for displacement degree of freedom	0.2 - 3.0
	Hour glass stiffness scaling factor for rotational degree of freedom	0.2 - 3.0
	Hour glass stiffness scaling factor for out-of-plane displacement degree of freedom	0.2 - 3.0

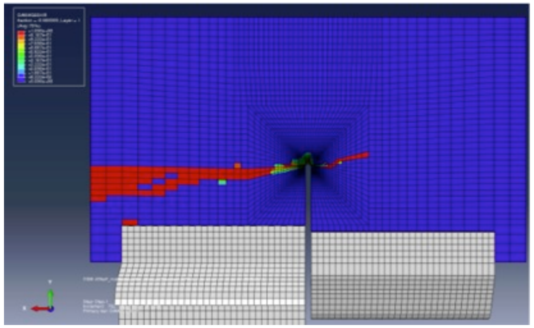
Scaling Hourglass Stiffness Cont

- Scaling $s\uparrow w$ caused solutions to fail prematurely
- Only scaled $s\uparrow s$ and $s\uparrow r$
- After scaling to the limits of the recommended value, not all stacking sequences converged
- After drastically increasing factors, convergence was achieved for most models
- Factors were selected based on a convergence study

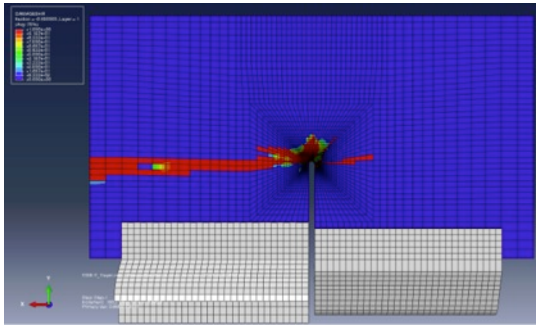
Damage Path Model



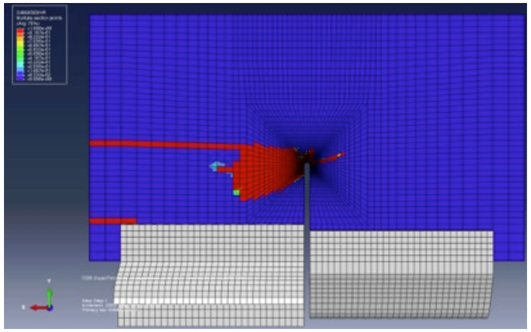
1 Layer – No SSF



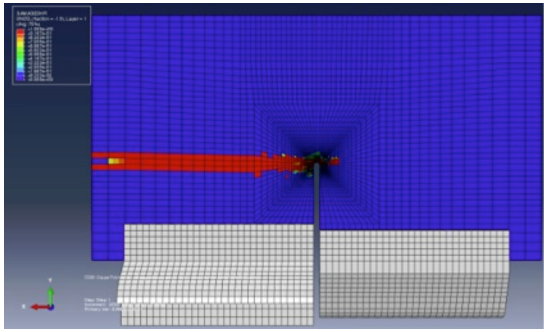
1 Layer – with SSF



2 Layer – VCCT



3 Layer – VCCT



Results Table: Explicit and Helius:MCT

Energy (Given)				
Combo	**MCT (N)**	**Exp. (N)**	**% Diff**	**Converge**
F	2330.83	1188	65.0	Y
N	2377.34	1689	33.9	Y
P	2598.69	1472	55.4	Y
AN	9785.4	5111	62.8	N
FP	9278.33	4005	79.4	Y
AR	7394.08	5899	22.5	Y

Instant Degradation (Default)				
Combo	**MCT (N)**	**Exp. (N)**	**% Diff**	**Converge**
F	1254	1188	5.4	N
N	1514	1689	10.9	N
P	1624	1472	9.9	N
AN	5182	5111	1.4	N
FP	4817	4005	18.4	N
AR	6528	5899	10.1	N

Instant Degradation (Given)				
Combo	**MCT (N)**	**Exp. (N)**	**% Diff**	**Converge**
F	1296.9	1188	8.8	N
N	1184.99	1689	35.1	N
P	1388.43	1472	5.8	Y
AN	4989.86	5111	2.4	N
FP	5104.25	4005	24.1	N
AR	6528.27	5899	10.1	N

Cohesive Zones (Given - Instant)				
Combo	**MCT (N)**	**Exp. (N)**	**% Diff**	**Converge**
F	713	1188	50.0	N
N	996	1689	51.6	N
P	838	1472	54.8	N

Abaqus/Explicit: Filter				
Combo	**Explicit (N)**	**Exp. (N)**	**% Diff**	**Converge**
F	1291	1188	8.3	Y
N	928	1689	58.1	Y
P	1158	1472	23.8	Y

Legend: Y = Yes, N=No, Exp. = Experimental Values, MCT= Helius:MCT results

F= 10% zeros, 20 ply ; P = 30% zeros, 20 ply ; N=50% zeros, 20 ply ; FP = 10% zeros, 40 ply ; AR = 30% zeros, 40 ply ; AN = 50 % zeros, 40 ply

Helius:MCT Results – Boeing Parameters (Energy Degradation)

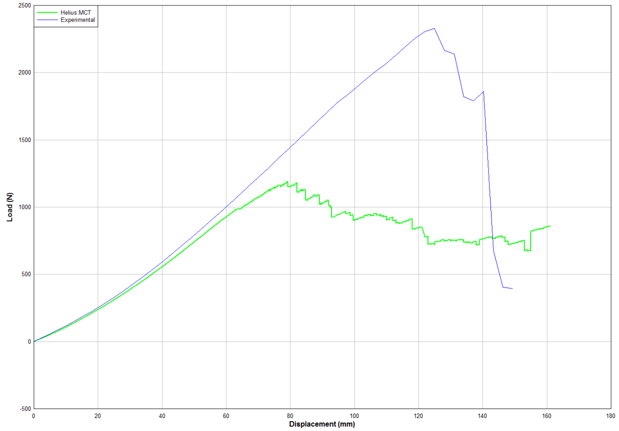


Fig 1. F Configuration

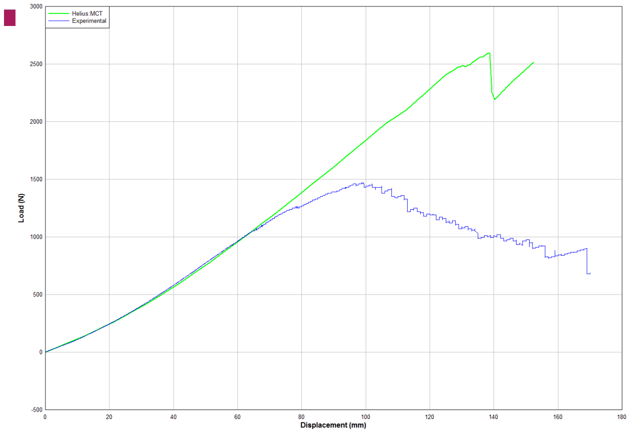


Fig 2. P Configuration

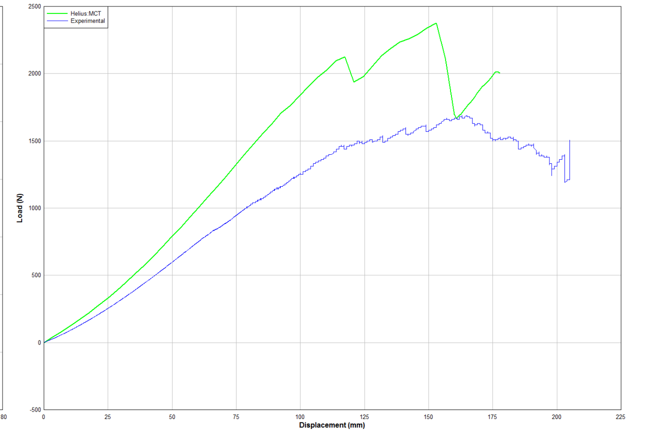


Fig 3. N Configuration

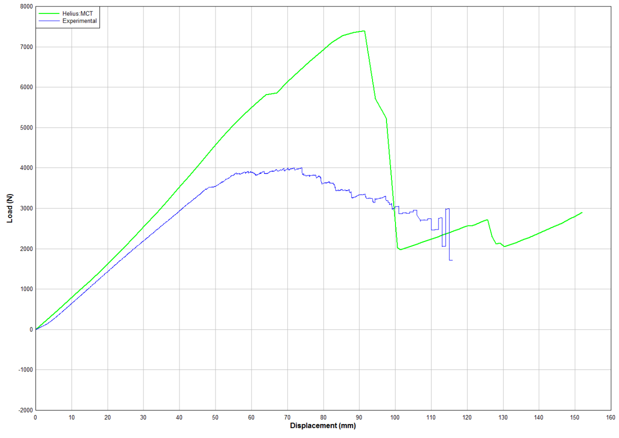


Fig 4. FP Configuration

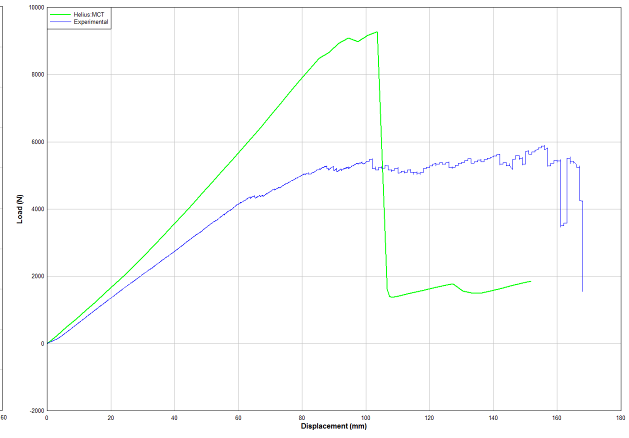


Fig 5. AR Configuration

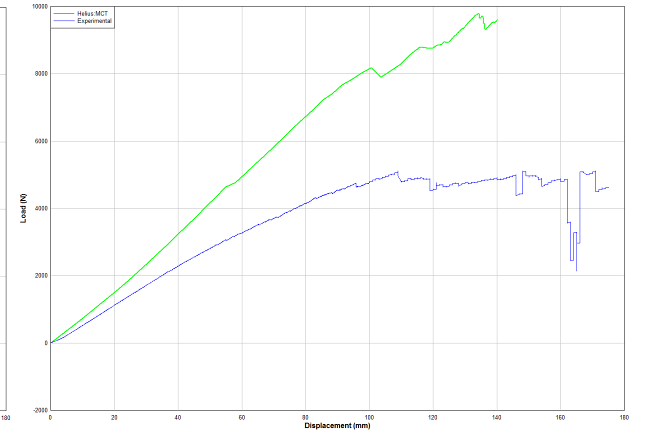


Fig 6. AN Configuration

A Center of Excellence
AMTAS
Advanced Materials in Transport Aircraft Structures

Helius:MCT Results – Boeing Parameters (Instant Degradation)

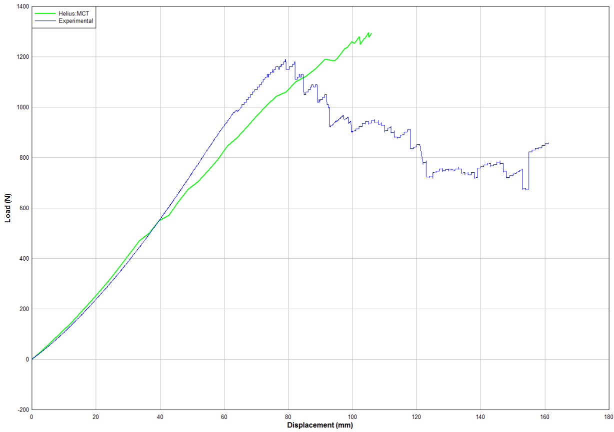


Fig 1. F Configuration

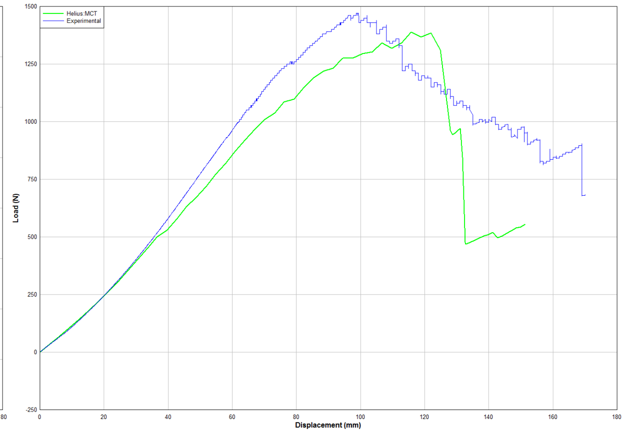


Fig 2. P Configuration

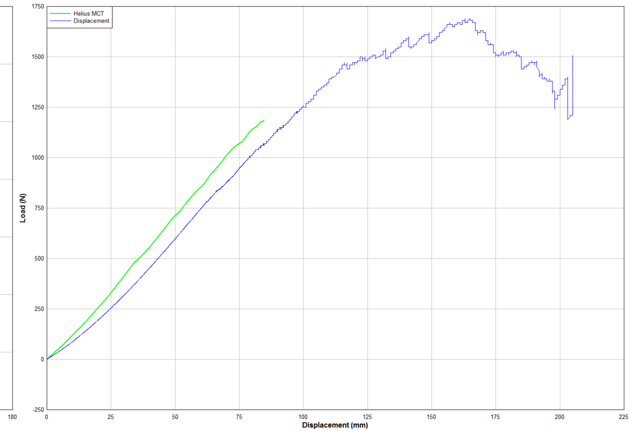


Fig 3. N Configuration

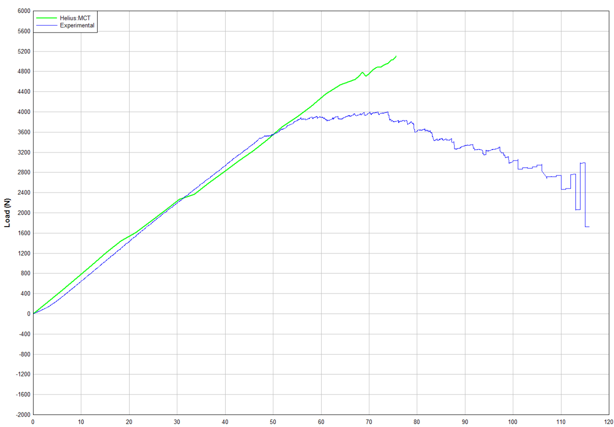


Fig 4. FP Configuration

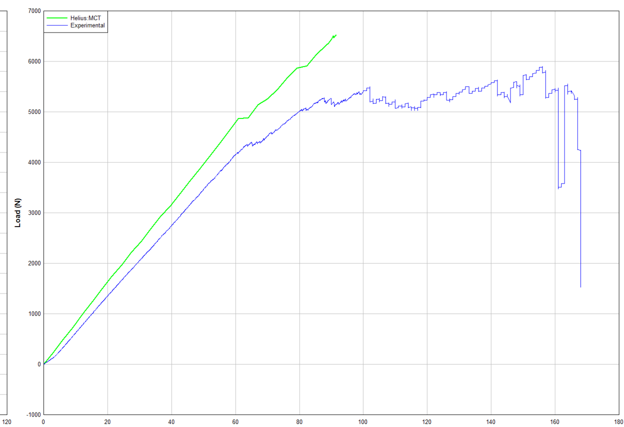


Fig 5. AR Configuration

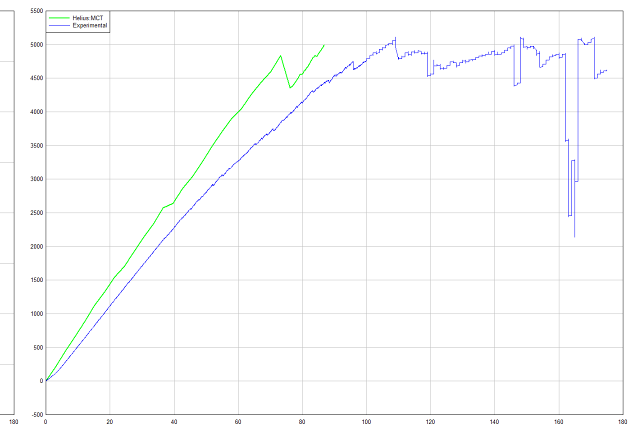


Fig 6. AN Configuration

Helius:MCT Results – Default Parameters

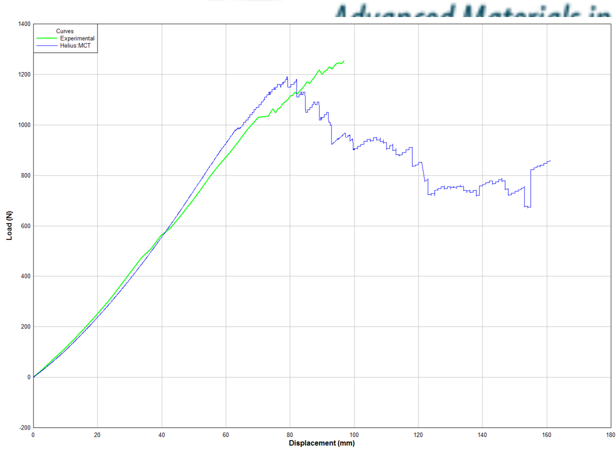


Fig 1. F Configuration

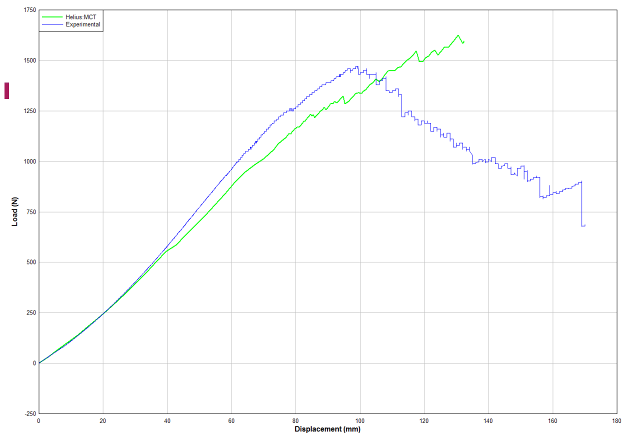


Fig 2. P Configuration

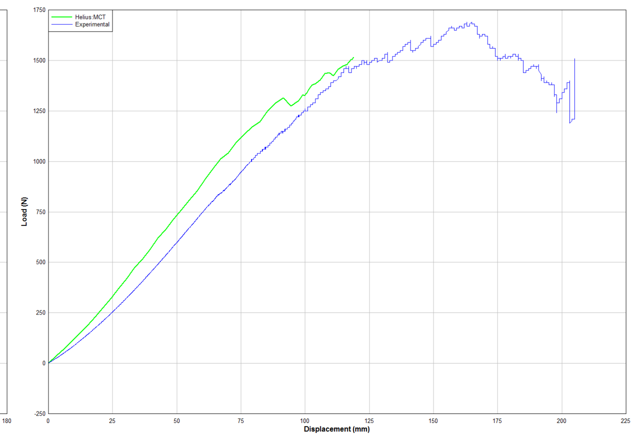


Fig 3. N Configuration

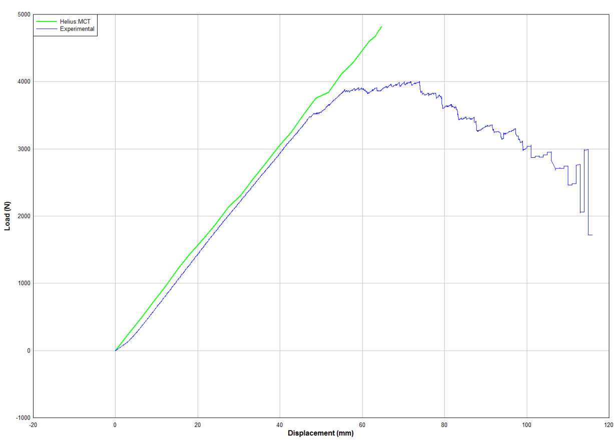


Fig 4. FP Configuration

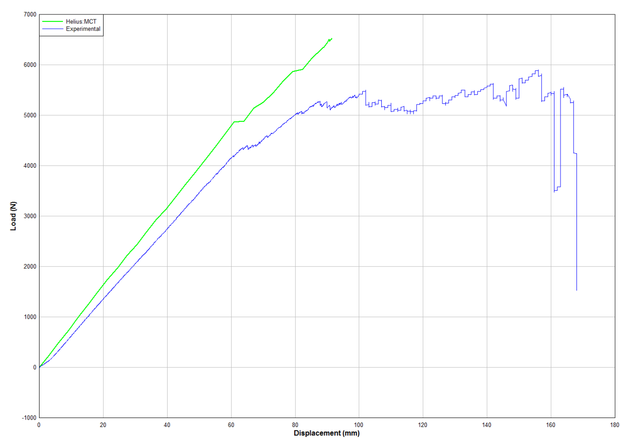


Fig 5. AR Configuration

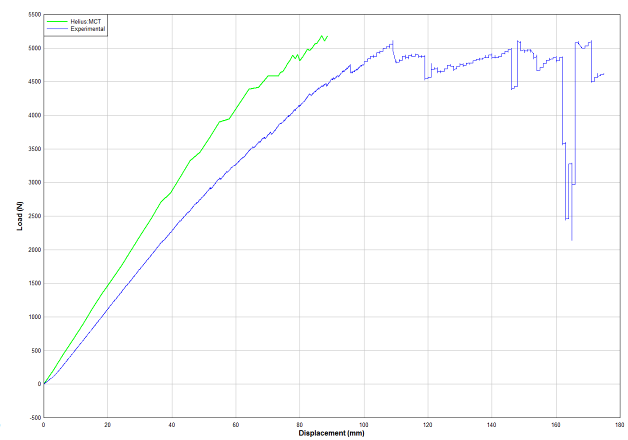
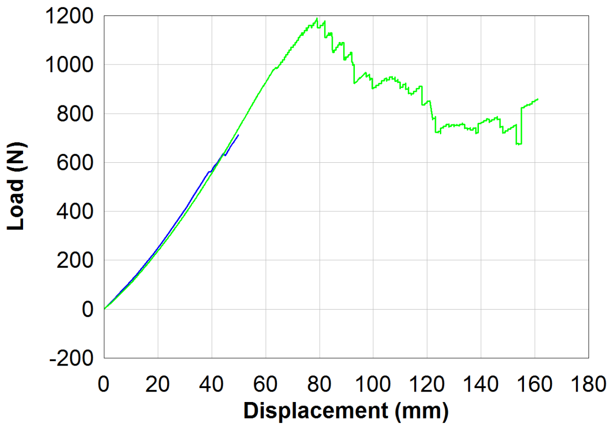
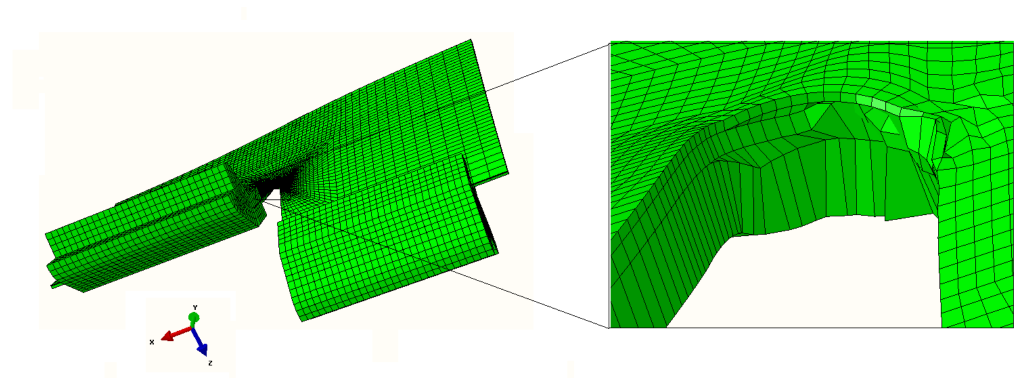
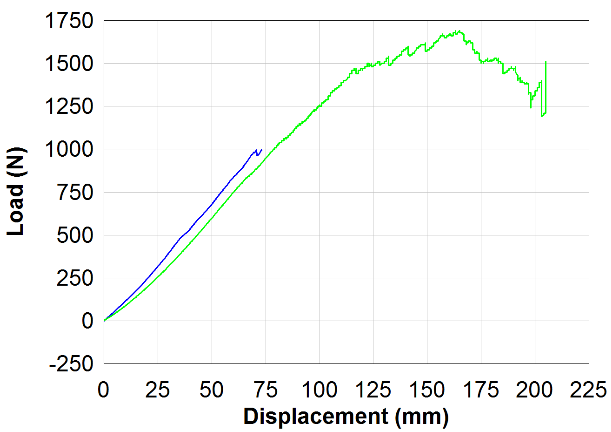


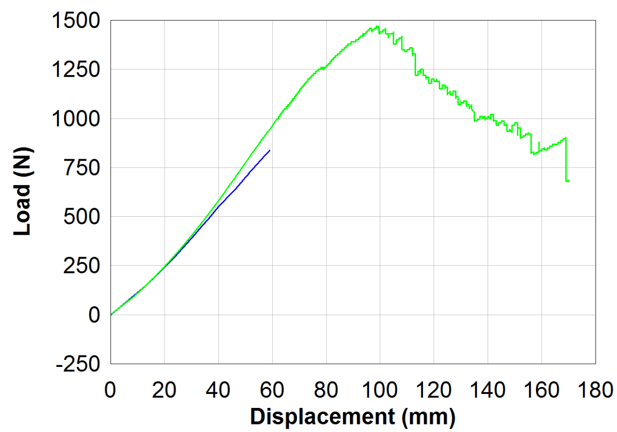
Fig 6. AN Configuration

Cohesive Zones in Helius:MCT



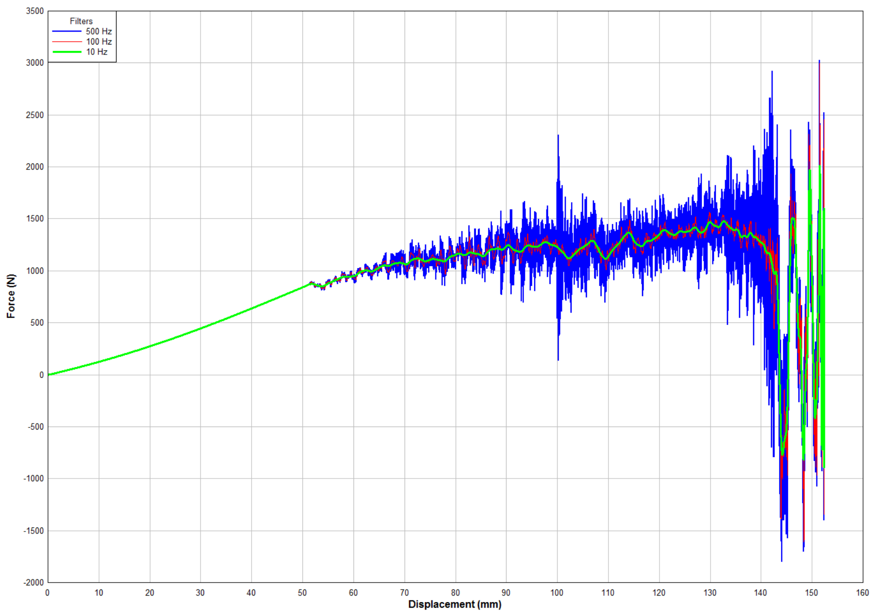






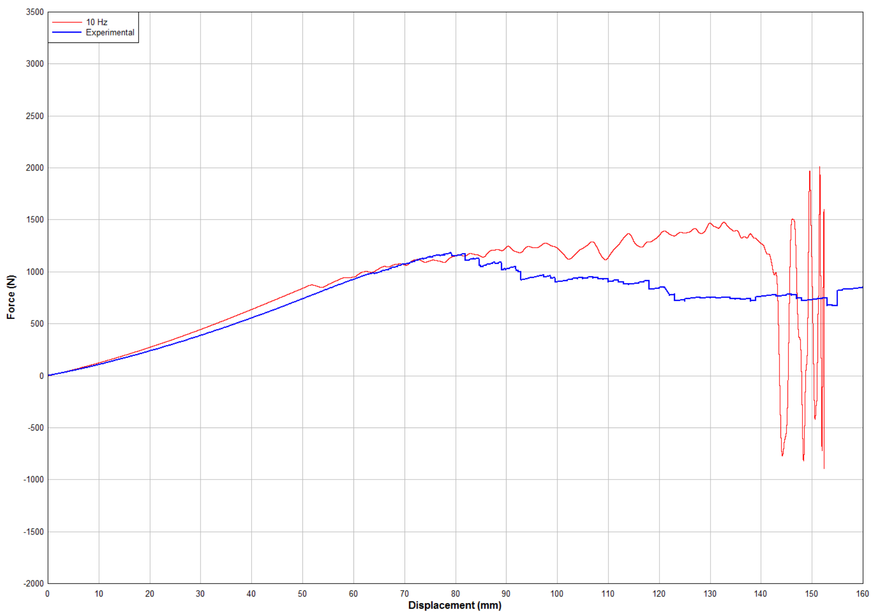
- Cohesive zone runs do not converge
- Deformation in cohesive zone areas can be observed but it is difficult to discern if this deformation is delamination

Filtering Results



Application of filters with varying cut-off frequencies for F-configuration

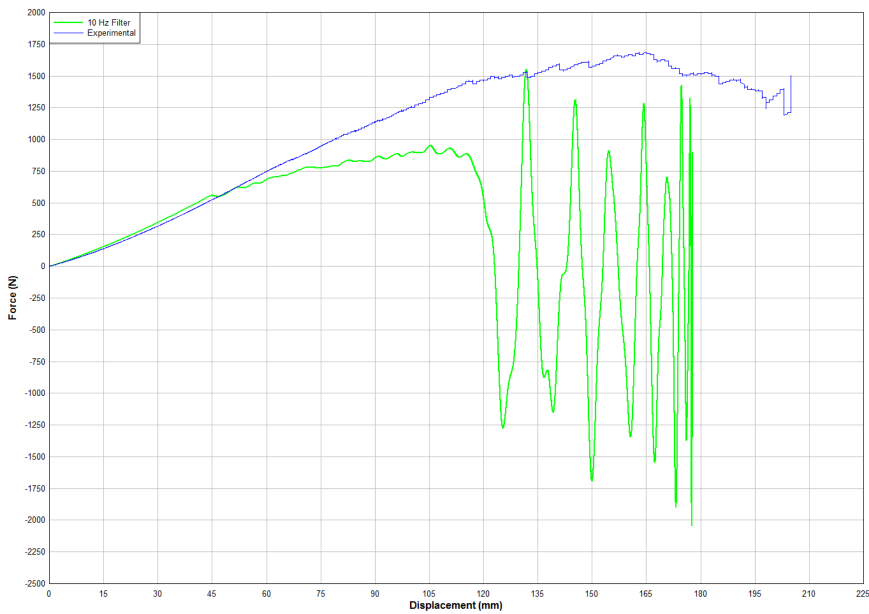
*10% zeros, 20 plies



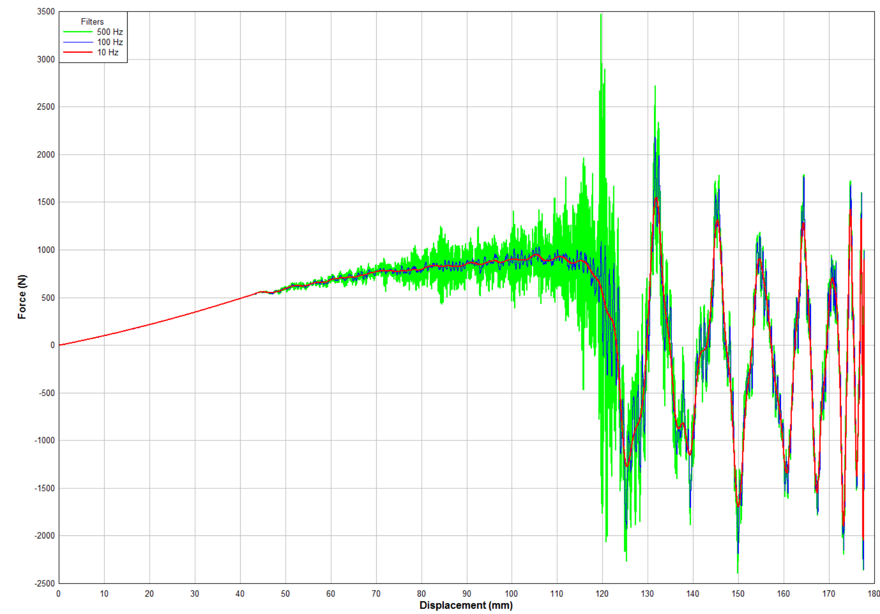
10 Hz cut-off filter compared to experimental results for F-configuration

Filtering Results

*50% zeros, 20 plies

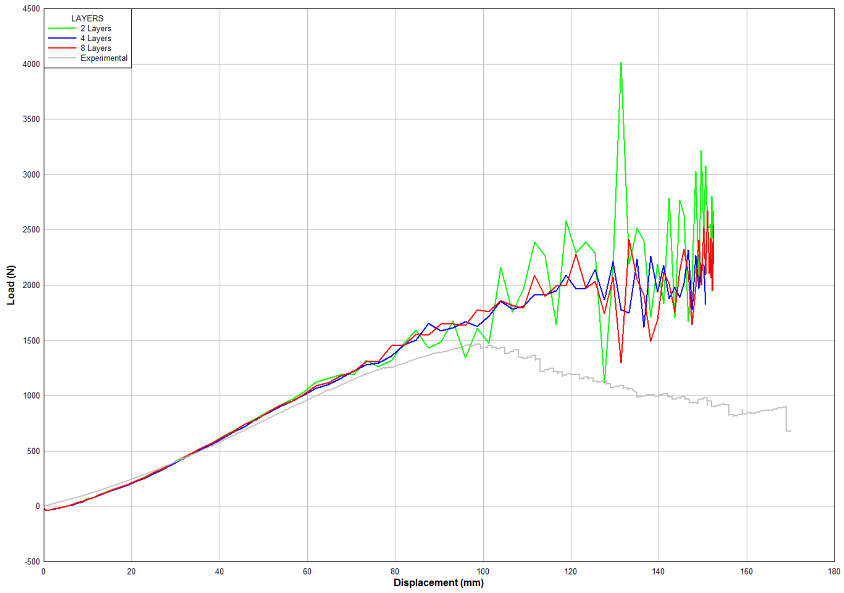


10 Hz cut-off filter compared to experimental results for N-configuration

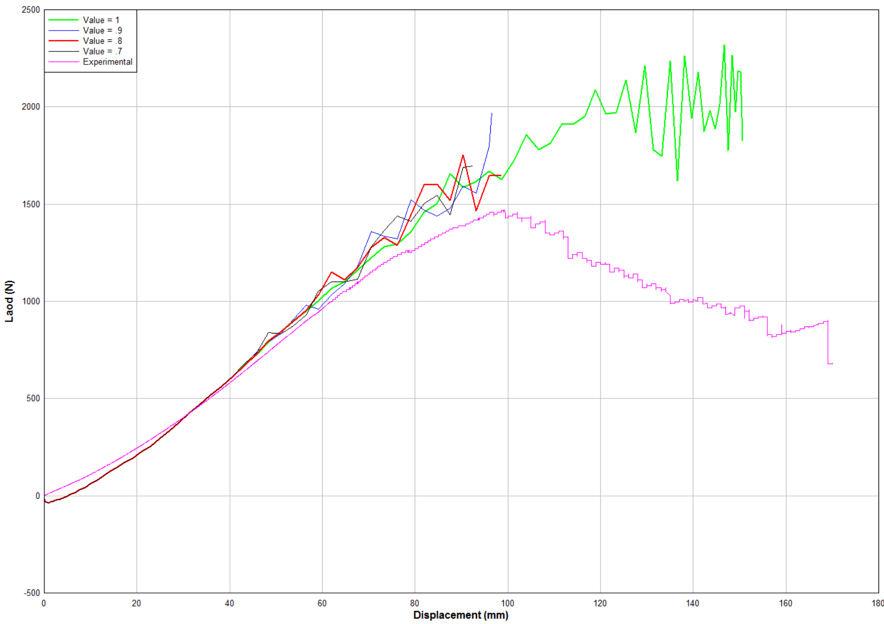


Application of filters with varying cut-off frequencies for N-configuration

More Multi-Layer Results



Multi-Layer Models : 30% zeros, 20 plies configuration



4 layer with varying degradation values: 30% zeros, 20 plies configuration

Abaqus/Explicit Solver Runtime

- Analyses are extremely long
  - the Explicit solver is only conditionally stable and requires an extremely small time step. Critical time step must considered:

$$\Delta t \leq \frac{2}{\omega_{max}} \leq \Delta t_{cr}$$

  - Need to maintain a Quasi-static state: $E_K \leq 0.1 \, E_I$

Multi-Layer Run Time Table

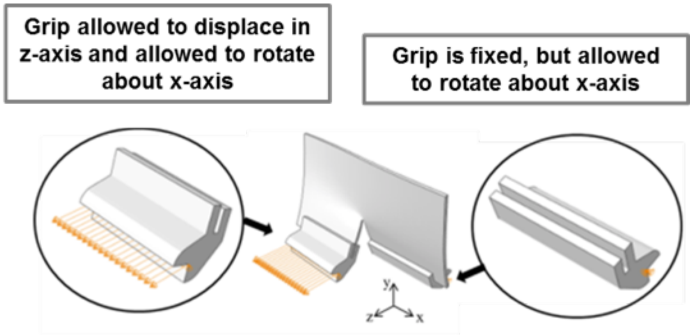
Table 1. Run Times for Quasi-static models.

Layers (ct.)	Run Time (hr)
2	354
4	672
8	585

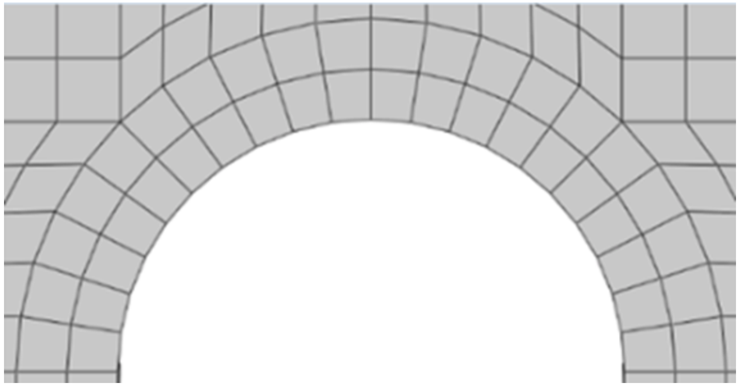
Evaluation of Abaqus Standard: Computational Model

- Solver basics:
  - Uses Newton-Raphson Technique to iterate to a converge solution for each time increment
  - Static equilibrium:

$$[K][U] = [R]$$



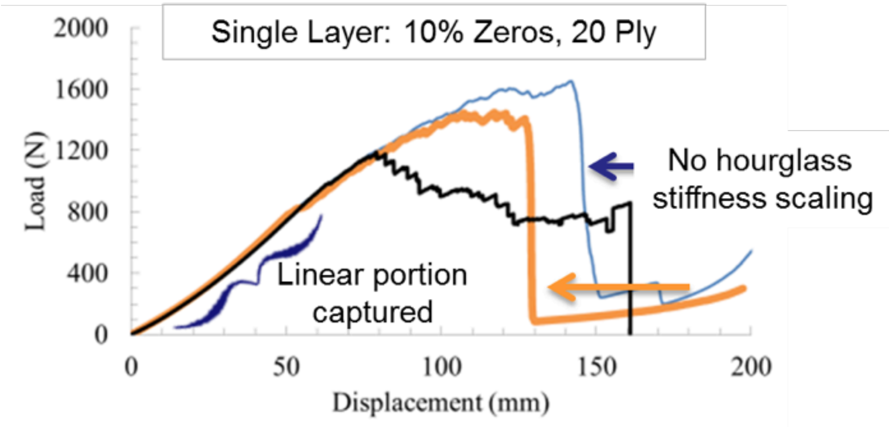
- Uses Hashin failure criteria
- Quasi-static analysis and Non-linear geometry turned on
- Panel: Continuum shell, reduced integration elements (SC8R)
- Grips: Continuum, 3-D, 8 node, reduced integration element (C3D8R)
- Boundary conditions implemented by grips
- Mesh Selection – 20 elements around notch tip, based on a linear elastic convergence study

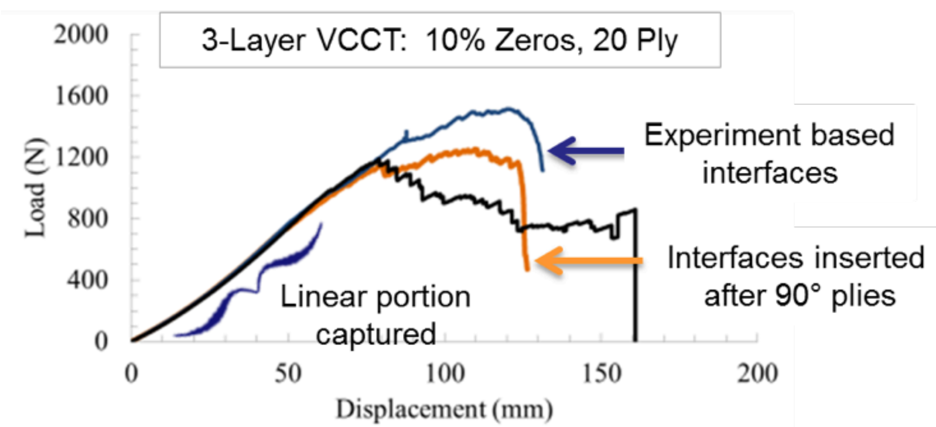


Evaluation of Abaqus Standard: Computational Model and Convergence Based Parameters

- Viscous Regularization Scheme (used in standard/explicit) helps with convergence
  - Viscosity coefficients for fiber compression , fiber tension , matrix tension , and matrix compression
  - Must be small with respect to the time increment,
  - Convergence trend at:
- Hourglass stiffness scaled to prevent severe element deformation (Standard only)
  - Three hour glass scaling factors for displacement degree of freedom rotational degree of freedom , and out-of-plane displacement degree of freedom
  - Scaling to recommended values didn't yield converged solutions for some stacking sequences in Standard
  - Needed to drastically increase factors, most models and stacking sequences converged
  - Converging trend at

Evaluation of Abaqus Standard: Single Element Layer and 3 Element Layer Delamination with VCCT Results



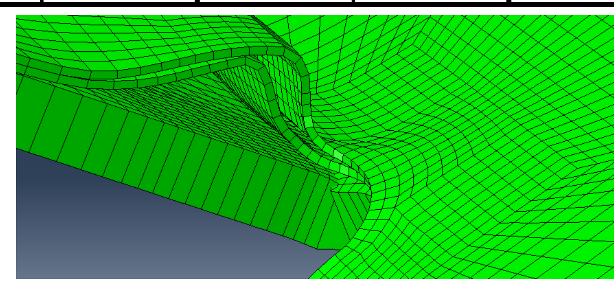


- All models effectively captured linear region
- FE models have stiffness factors scaled high
- FE material response is similar, but 3-layer VCCT models capture experiment behavior better
- Not all models revealed a clear max

Evaluation of Abaqus Standard: Summary of Results

- Benefit: Standard predicts max load within 20% of experiments
- Major Challenges:
  - Implicit analysis fails to converge without excessive stiffness factors
  - After the use of excessive stiffness factors, some models still fail to converge
- Suggestion:
  - Accuracy can be improved by changing VCCT interfaces – but no rational for it
  - Modify convergence parameters

		FEA % Difference from Average Experimental Maximum Load					
	% Zero	1 Element Layer - No Scaled Stiffness Factors (SSF)	1 Element Layer - with SSF	2 Element Layer VCCT with SSF - Interface from Experiments	2 Element Layer VCCT with SSF - Interface before 90° plies	3 Element Layer VCCT with SSF - Interface from Experiments	3 Element Layer VCCT with SSF - Interface before 90° plies
20 PLY	10%	31%	15%	25%	20%	20%	**-1%**
	30%	DNC	19%	21%	21%	DNC	**18%**
	50%	**-16%**	**-16%**	-23%	-23%	DNC	DNC
40 PLY	10%	45%	22%	DNC	18%	DNC	**17%**
	30%	DNC	3%	**-2%**	-7%	42%	22%
	50%	**-3%**	-5%	-4%	-4%	DNC	DNC



Today’s Topics

- Experimental results: Out-of-plane shear
- Evaluation of Abaqus Standard results
- Evaluation of Helius MCT results
- Evaluation of Abaqus Explicit results

A Center of Excellence
AMTAS
Advanced Materials in Transport Aircraft Structures

Abaqus/Standard with Helius:MCT

- Helius:MCT was utilized for its recognized convergence capabilities and fast solver algorithm for out-of-plane bending
- Solver basics: analyzes the composite based on its constituents as well as a whole:

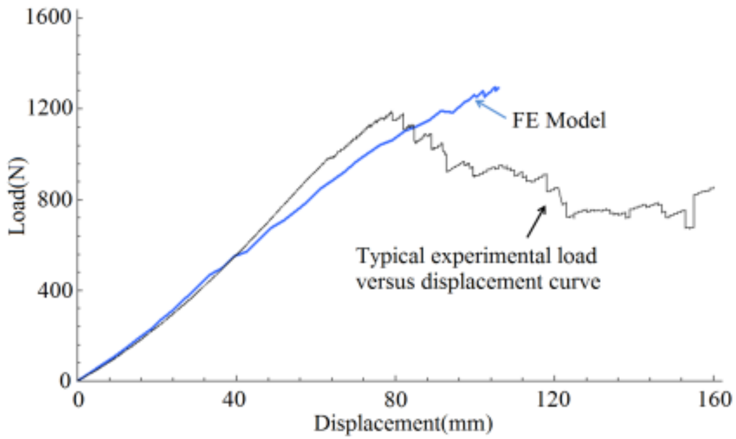
Volume fractions

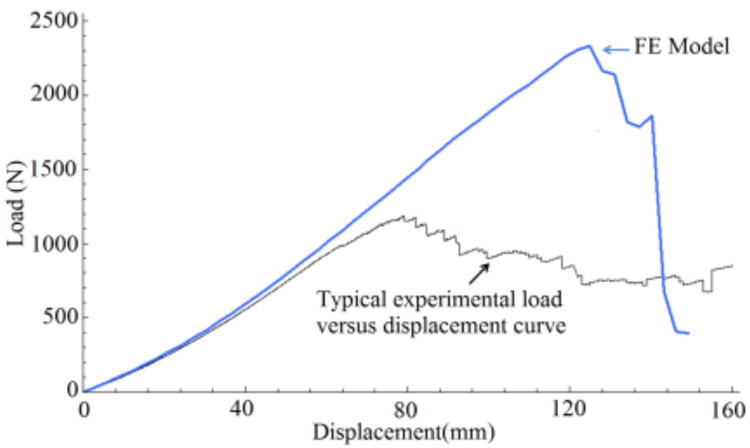
$$\sigma^c = \phi_f \sigma^f + \phi_m \sigma^m$$

Average stress of composite, fiber, and matrix respectively

- Method:
  - Adapt input file to include Helius:MCT solver
  - Use default parameters, instant degradation parameters, energy degradation parameters
  - Apply cohesive zones (CZ)

Abaqus/Standard with Helius:MCT





- Representative of all trials and configurations, including with CZ
- Benefits:fast solver: runtime < 10hrs
- Major challenges:
  - Convergence
  - Accuracy in certain situations
- Suggestions
  - Shows promise if convergence occurs, try different energy parameters or degradation values
  - Possible changes may occur in the future to better the solver: Autodesk ownership

Today's Topics

- Experimental results: Out-of-plane shear
- Evaluation of Abaqus Standard results
- Evaluation of Helius MCT results
- Evaluation of Abaqus Explicit results

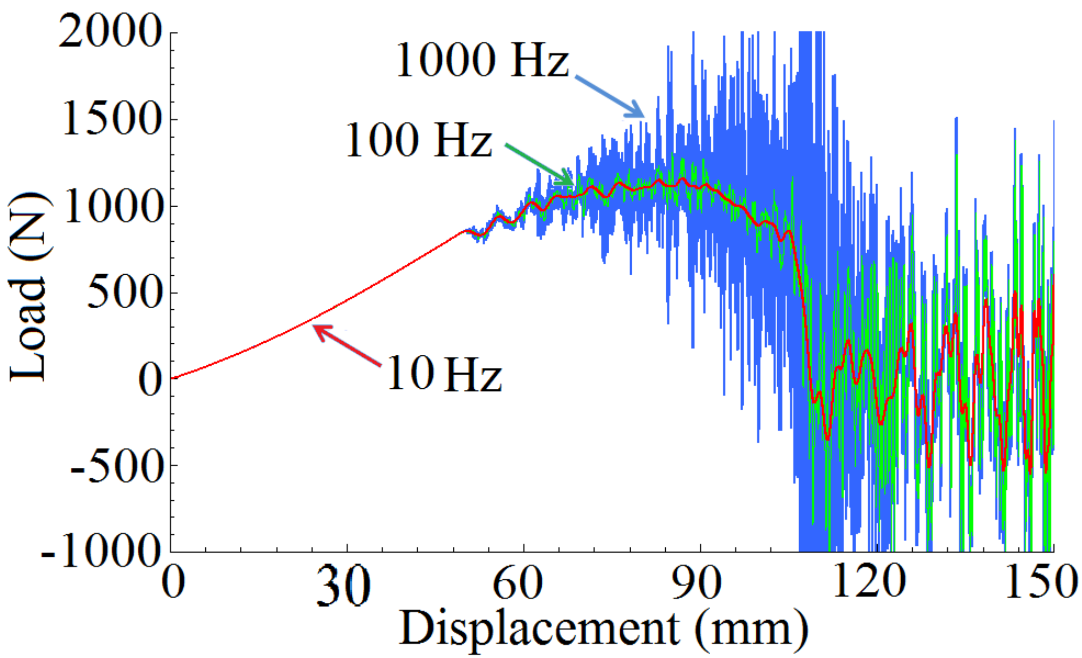
Abaqus/Explicit Analysis

- Why use explicit: implementation of element deletion and better convergence
- Solver basics:
  - Analysis used an explicit, dynamic solver:

$$[M][\ddot{U}] + [C][\dot{U}] + [K][U] = [R]$$

  - Central difference method for enhanced convergence: hope to overcome the issues present in Abaqus/Standard
  - Hashin damage criteria
- Determination of quasi static state
  - Varied total time until a majority of analysis was quasi static: kinetic energy < 10% internal energy
  - Total time considered (seconds): 0.25, 0.50, 1.00, 2.00, 4.00, 6.00, 8.00
  - 8 seconds chosen as total time increment

Abaqus/Explicit Analysis: One Layer Results

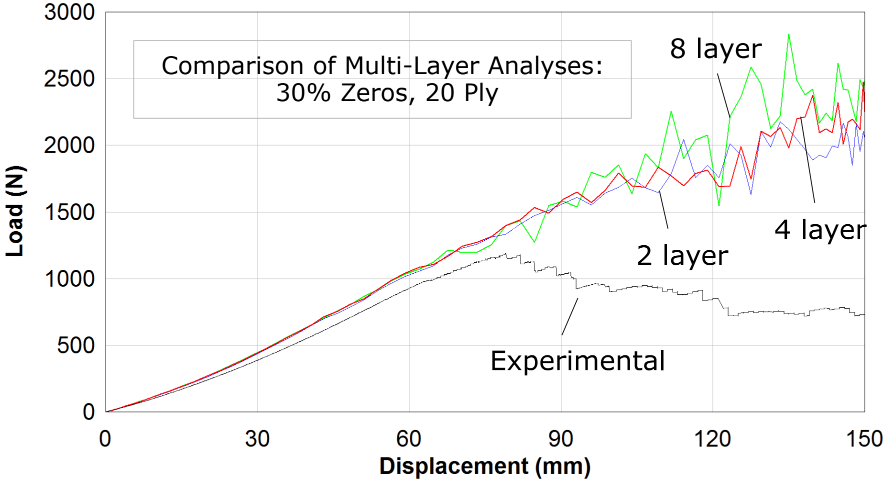


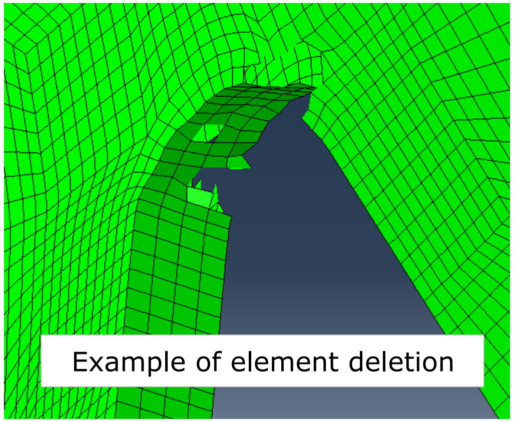
- Benefits: convergence in most cases
- Major challenges:
  - Extreme amounts of noise
  - Extremely long runtime
- Suggestions
  - Filtering the data
  - Implementing more layers

Abaqus/Explicit Analysis: Implementation of Multiple Layers and VCCT

- Dilemma: element deletion is not occurring
- Solution: create more layers so that the deletion criteria is met more readily
- Methods:
  - Create 2,4, and 8 layer models.
  - Varying degradation coefficient: 1.0,0.9,0.8,0.7
  - Implement VCCT

Abaqus/Explicit Analysis: Implementation of Multiple Layers





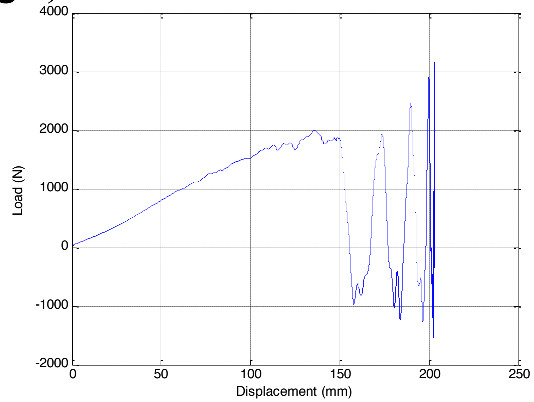
- Benefits: eliminate distorted elements
- Major Challenges:
  - Convergence
  - Extremely long run times
- Suggestion: not much can be gained overall from implementing multiple layers

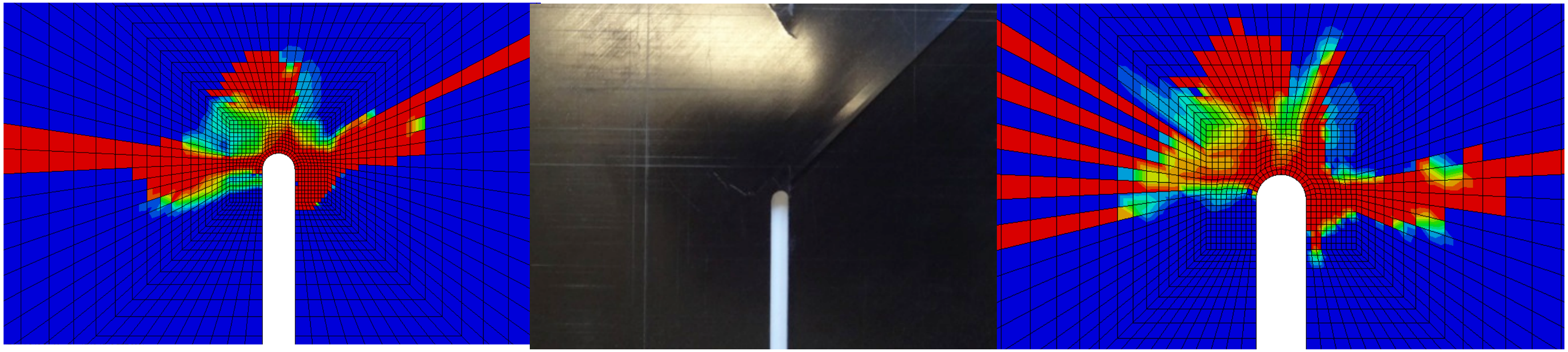
Conclusions for Explicit Analysis

- Analyses are extremely long
- Analyses are inherently prone to noise during extreme deformations or accelerations. How do we appropriately filter this noise?
- Convergence is not guaranteed and element deletion may not always be something we can take advantage of.

Future Work-Explicit VCCT

- Revisited VCCT with explicit
- Showed better results for damage path (45° damage)
- Showed better stress strain curve shape
- Over predicts maximum load
- Longer solve time: ~90 hours
- Difficult convergence
- Future work: refine placement of cohesive layers

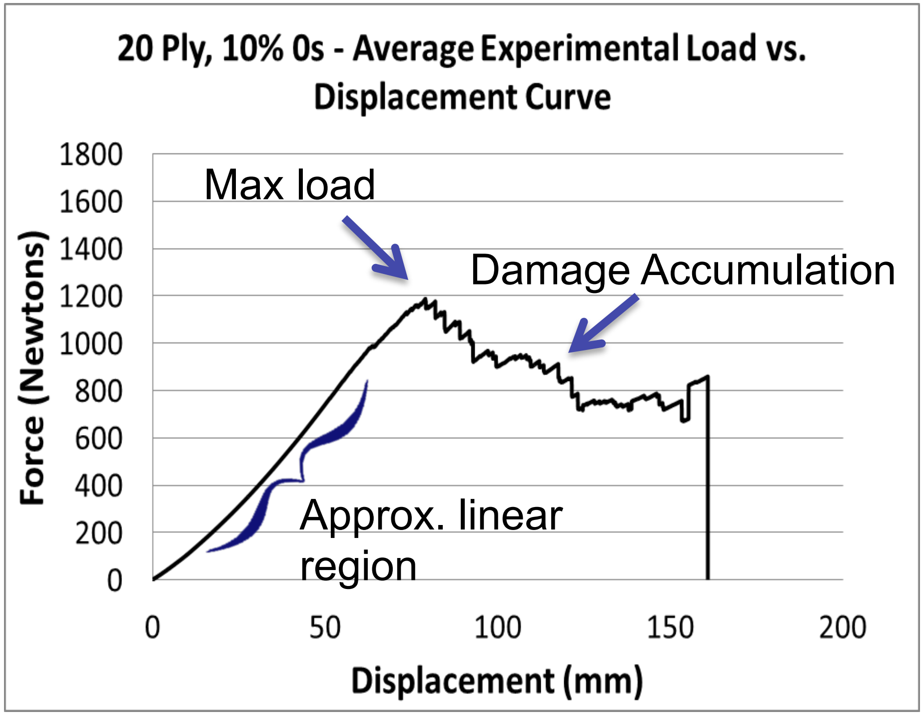


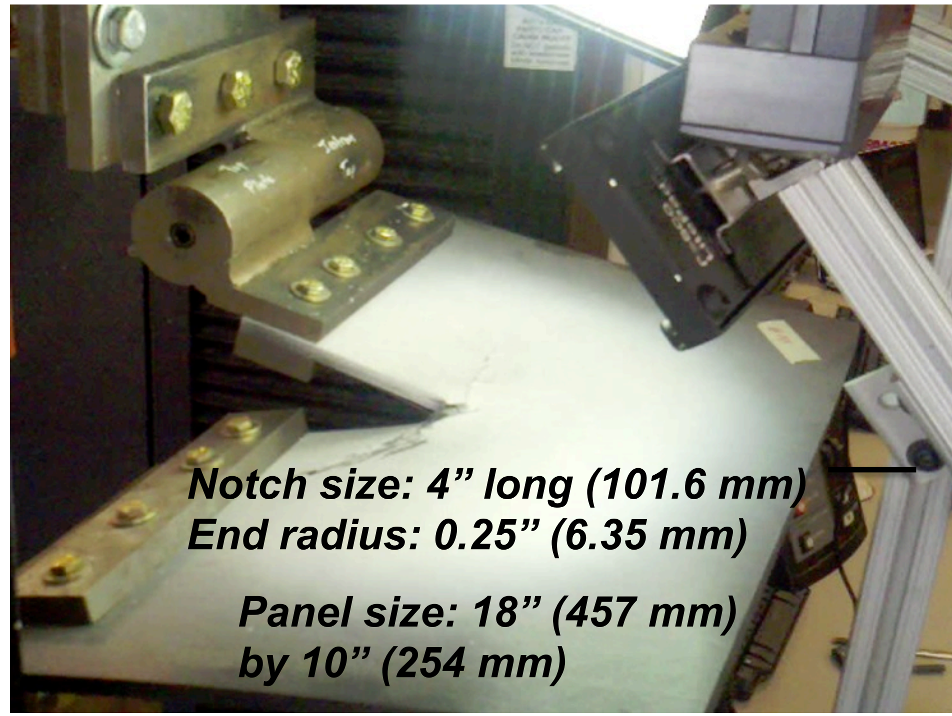


Future Work-Energy Sensitivity

- Previous work by Ludeman showed significance in energy parameters for quasi isotropic layups with a 2^{10-6}
- First attempts at a thick quasi-isotropic layup yielded no energy significance
- All 90s showed significance of matrix energies
- Attempting to fully recreate Ludeman's runs with 2^{10-4} fractional factorial

Project Overview

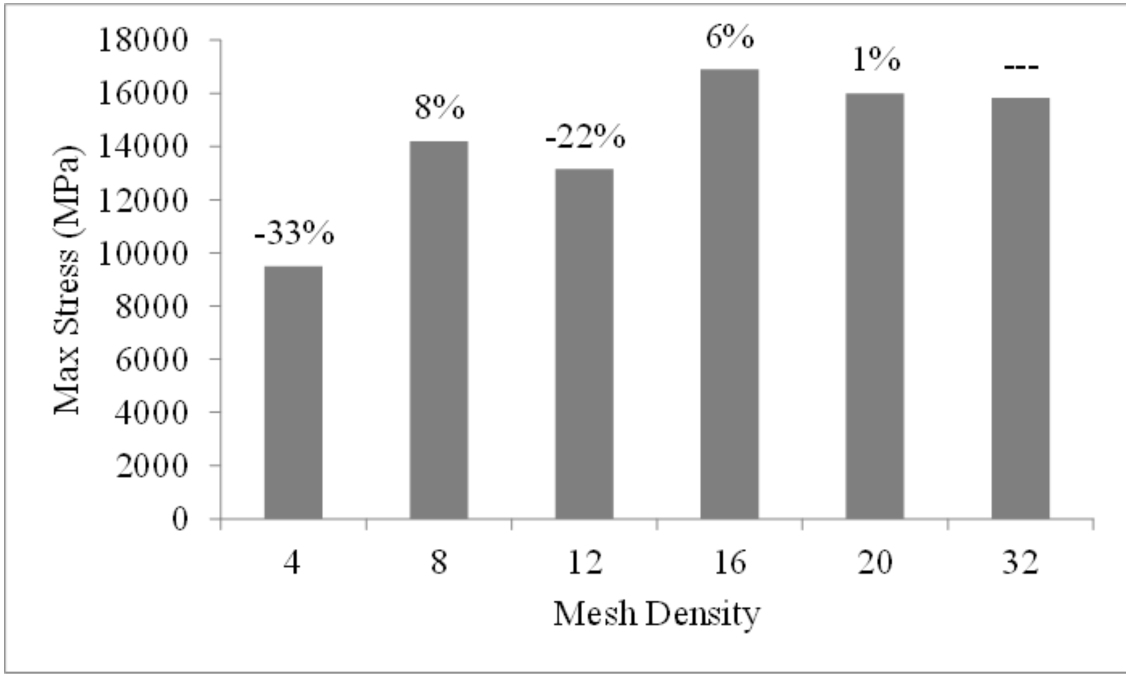




- Edge-notched CF panels displaced to maximum load
- 20 and 40 lamina thick panels with three lay ups: 10%, 30%, & 50% 0° plies
- Metrics: Applied displacement and applied load

Improvements for Abaqus Explicit-Mesh Study

- Hashin criteria mesh dependent in Abaqus
- Mesh convergence study to determine most accurate mesh
- Ran simulations in linear elastic region to compare stress values of different meshes
- Selected coarsest mesh showing converged stress value
- Plot shows percent deviation from previous mesh



Sensitivity Study- Significance Results

- Normal probability plot of 50% 0° explicit results shown to the right
- Deviation from linear line indicates significance
- Further deviations suggest more significant effects
- In example: Xt shows more significance than Xc