MAE 646. Advanced Mechanics of Composite Materials
Spring 2015 – CRN

Instructor: Dr. Barbero
Schedule:
Office hours:

Textbook:

Finite Element Analysis of Composite Materials Using Abaqus, CRC (2010), Ever J. Barbero. ISBN 1-4200-5433-3

SOFTWARE in G78: (no prerequisite knowledge required)
Abaqus 6.10, Intel Visual Fortran 11.1, MATLAB, Python 2.7.

Prerequisites:

Some knowledge of composite materials or consent

Course Objective:

The objective of this course is to present advanced analysis techniques used to support the design of composite structures. The course covers those topics overlooked during preliminary design courses such as MAE 446.

During preliminary design, the material is selected and the structure is roughly sized to carry the loads; many design aspects are delayed for the final design stage (for example, selection of the optimum stacking sequence that minimizes edge effects). Furthermore, many design allowables, such as notched strength, creep rate, and so on, can be determined by advanced modeling techniques of a few candidate materials. These values are then supplied as data for the preliminary design effort. On the other hand, refined computations of deflections, stress, strength, and buckling loads can only be done using finite element analysis. FEA of composite structures includes many aspects that set it apart from standard FEA, thus requiring some attention as part of this course.

References:

1. Barbero, E. J., Introduction to Composite Materials Design—Second Edition, CRC, 2010.
2. J. Aboudi, Mechanics of Composite Materials, Elsevier, Amsterdam, 1991 (available at Evansdale Library EAM8445)
3. J. N. Reddy, An Introduction to CONTINUUM MECHANICS With Applications.
4. Mase, T. G., Smelser, R. E., and Mase, G. E., Continuum Mechanics for Engineers-Third Edition (2010), ISBN 1420079158

Outline

1. 3D Mechanics of Composites (weeks 1-2)
1.1. Introduction to composite materials
1.2. Tensor Analysis, Contracted Notation, Transformations (computer assignment)
1.3. 3D Constitutive Equations and Micromechanics (computer assignment)
1.4. 3D Failure Criteria

2. Finite Element Modeling (week 3-4)
2.1. Review FEM
2.2. Mesh Generation
2.3. Loads
2.4. Boundary Conditions-Symmetry-Periodicity
2.5. Constraint Equations
2.6. Introduction to Abaqus/CAE

3. Stress Analysis (weeks 5-6)
3.1. Conventional Shell Elements
3.2. Continuum Shell Elements
3.3. Deformations, Contour, and Deformed Geometry Plots
3.4. Layer-by-layer stresses
3.5. Effect of stacking sequence (computer assignment)

4. Buckling and Post-Buckling (weeks 7-8)
4.1. Perfect System, Bifurcation, Post-buckling Path (computer assignment)
4.2. Imperfect System
4.3. Nonlinear Analysis (computer assignment)

5. Free Edges (week 9)
5.1. Approximate Equations (computer assignment)
5.2. 3D Solid Elements
5.3. Finite Element Analysis (computer assignment)

6. Computational Micromechanics (week 10)
6.1. Theory
6.2. Implementation
6.3. Introduction to Python

7. Viscoelasticity (weeks 11-12)
7.1. Fitting of Experimental Data (computer assignment)
7.2. Laminated composite creep properties from matrix creep data (computer assignment)
7.3. Viscoelastic material in FEM codes (computer assignment)

8. Delaminations (week 13-14)

8.1. Virtual crack closure
8.2. VCC implementation (computer assignment)
8.3. Decohesion
8.4. Decohesion implementation (computer assignment)

To be updated

Week	**Date**	**Section**	**Subsection**	**Example**	**Hmwk due**	**Observations**
1	1/10	1	Intro			
1	1/12	1	Intro			
2	1/17	1	1.10	1.3		
2	1/19	1	1.13		1	
3	1/24	1	1.4			
3	1/26	2	2.1.1-.3		2	
4	1/31	2	2.1.4-.10	2.2		
4	2/2	3	Intro		3	
5	2/7	3	3.1			
5	2/9	3	3.1			
6	2/14	3				
6	2/16	3				
7	2/21	4	Intro			
7	2/23	4	4.1	4.2		
8	2/28	4	4.2	4.3		
8	3/1	5			5	
9	3/6	5				
9	3/8	5				
10	3/13	6				
10	3/15	6				
11	3/20	8	Ch.10			
11	3/22	8	Ch.10			
12	3/27	-				
12	3/29	-				
13	4/3	6				
13	4/5	6				
14	4/10	7				
14	4/12	7				
15	4/17	P				
15	4/19	P				
16	4/24	P				
16	4/26	P				

Last year projects

Team Acronym	**Title**	**Participants**	**Prereq.**	**Mid-term deliverable**
MatSci	Magneto-resistive/piezo-electric composite	Campo, Muchenick, Omar	Ch. 3	Mechanical model
Ugen I	UGEN Ply-discount E-Glass	Ibraheem-Chada	Ch. 3	CLT UGEN
Ugen II	UGEN Ply-discount Carbon-Epoxy	Dittenber-Dispenette	Ch. 3	CLT UGEN
Textile	UMAT textile fabric	Eckman-Peil	Ch. 3	Matlab model
Damage I	Abaqus damage E-Glass	Gosh-Praveen-Wattick	Ch. 8	Damage initiation
Damage II	Abaqus damage Carbon-Epoxy	Polina-Qureshi	Ch. 8	Damage initiation
G78	Basket weave	Montejo-Wong	Ch. 6, and 8	Plain weave laminate moduli

Grading: % distribution subject to change

Homework	10%	Individual
Computer assignments	40%	Team
Midterm	20%	Individual
Project	30%	Team
Final	0%	TBA

Teamwork:
Teams of maximum 2 students will be formed. Teams, not individuals, will submit computer assignments and projects. Although help and discussions are encouraged among the class at large, various teams are forbidden to work together and/or present essentially similar work. Similarities in their work will be penalized. Innovative content and presentation quality will be rewarded.

Computer usage: FEM will be done with Abaqus and other software as needed. Code programming can be done either with MATLAB, Python, or FORTRAN. Plots must be computer generated. LaTeX or MSWord must be used for typed reports. All the necessary software is available in G78. E-mail and e-campus will be used for team communications. Other software is discouraged unless agreed in class. Learning the use of software is a lifelong learning ability that you must acquire on your own. All these applications have tutorials and online help for that purpose. Abaqus Student Edition is available for download free of charge to students and faculty of academic institutions in the U.S. https://swym.3ds.com/#community:73

NO AUDIT STUDENTS ALLOWED, WITHOUT EXCEPTION.

Updated: December 21, 2014