Course in FEM – ANSYS Classic

Geometric modeling

FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

Modeling Programme for Lesson:

- Modeling considerations
- Element Type
- Real Constants
- Material Properties
- Sections
- Geometry/Modeling
 - WorkPlane & Coordinate systems
 - Keypoints
 - Lines
 - Areas
 - Volumes
- Meshing

FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg Geometric modeling

BUILD THE MODEL

Review

Interpolation: A computation of an approximate value to a function on the basis of another function given by values in two outer points to an interval.

In the element method the values of the displacements $u_1, u_2, ...$ in the element nodal points are such points. These known values are degrees of freedoms in the element model and are computed/approximated by the solution of the system of algebraic system of equations.

Interpolation is the essence of the FE method. Using sufficiently fine element distribution with linear shape functions it is possible to model even relatively complex problems (geometrically as well as due stress state) applying simple interpolation functions.

Review

- Equilibrium for nodal forces and -moments is satisfied.
- Compatibility is satisfied in FE nodes.
- Equilibrium is not satisfied across the element boundaries.
- Compatibility is not necessarily satisfied across element boundaries. For the triangular and the rectangular element compatibility is satisfied as the element sides remain straight under deformation.
- Equilibrium is not satisfied for the individual element (due to the weak formulation – integral form).
- Compatibility is satisfied for the individual element, i.e. the displacement field must be continuous. This is automatically achieved by a proper formulation of the element shape functions, i.e. polynomial formulation.

- As you begin your model generation, you will (consciously or unconsciously) make a number of decisions that determine how you will mathematically simulate the physical system:
 - What are the objectives of your analysis?
 - Will you need to vary/modify model data?
 - Will you need to change the geometric topology of the model, e.g. add holes to the model?
 - Will you model all, or just a portion, of the physical system?
 - How much detail will you include in your model?
 - What kinds of elements will you use? How dense should your finite element mesh be?
- In general, you will attempt to balance computational expense (CPU time, etc.) against precision of results as you answer these questions.
- The decisions you make in the planning stage of your analysis will largely govern the success or failure of your analysis efforts.

- Linear or Higher Order Elements
- Take Advantage of Symmetry
 - The axis of symmetry *must* coincide with the global Cartesian Y-axis.
 - Negative nodal X-coordinates are not permitted.
 - The global Cartesian Y-direction represents the axial direction, the global Cartesian X-direction represents the radial direction, and the global Cartesian Z-direction corresponds to the circumferential direction.
 - Your model should be assembled using appropriate element types:
 - For axisymmetric models, use applicable 2-D solids with KEYOPT(3) = 1, and/or axisymmetric shells. In addition, various link, contact, combination, and surface elements can be included in a model that also contains axisymmetric solids or shells. (The program will not realize that these "other" elements are axisymmetric unless axisymmetric solids or shells are present.)
- How Much Detail to Include
- Appropriate Mesh Density

▲ Preferences for GUI Filtering	
[KEYW][/PMETH] Preferences for GUI Filtering	
Individual discipline(s) to show in the GUI	
	🗖 Structural
	🗖 Thermal
	🗖 ANSYS Fluid
	FLOTRAN CFD
Electromagnetic:	
	🗖 Magnetic-Nodal
	🗖 Magnetic-Edge
	🗖 High Frequency
	🗖 Electric
Note: If no individual disciplines are selected they \ensuremath{w}	ill all show.
Discipline options	
	• h-Method
	O p-Method Struct.
	O p-Method Electr.
or 1	Cancel

Characterization of problem



- The ANSYS program does not assume a system of units for your analysis.
- Units must however be consistent for all input data.

Geometric modeling

BEAM CIRCUit COMBINation PIPE CONTACt FLUID HF (High Frequency) **HYPERelastic INFINite INTERface** LINK MASS MATRIX

MESH Multi-Point Constraint PI ANF PRETS (Pretension) SHELL SOLID SOURCe SURFace TARGE TRANSducer USER VISCOelastic (or viscoplastic)

FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

Main Menu> Preprocessor> Element Type> Add/Edit/Delete



▲Library of Element Types			×
Library of Element Types	Structural Mass Link Beam Pipe Rigid Solid Shell	3D mass 21 3D mass 21	
Element type reference number	2	_,	
OK Apply	Cancel	Help	
The ANSYS element library of more than 150 different elem	contains ent types		
Each element type has a union a prefix that identifies the ele	que number and ment category		_ ET,1,BEAM4 ET,2,SHELL

- Many element types have additional options, known as KEYOPTs, and are referred to as KEYOPT(1), KEYOPT(2), etc. e.g.:
 - KEYOPT(9) for <u>BEAM4</u> allows you to choose results to be calculated at intermediate locations on each element
 - KEYOPT(3) for <u>SHELL63</u> allows you to suppress extra displacement shapes

Real Constants



Computational Mechanics, AAU, Esbjerg

Real Constants

- Element real constants are properties that depend on the element type, such as cross-sectional properties of a beam element
 - e.g. real constants for <u>BEAM3</u>, the 2-D beam element, are area (AREA), moment of inertia (IZZ), height (HEIGHT), shear deflection constant (SHEARZ), initial strain (ISTRN), and added mass per unit length (ADDMAS).
- Not all element types require real constants, and different elements of the same type may have different real constant values.

Real Constants

 For line and area elements that require geometry data (cross-sectional area, thickness, diameter, etc.) to be specified as real constants, you can verify the input graphically by using the following commands in the order shown:

> Utility Menu> PlotCtrls> Style> Size and Shape Utility Menu> Plot> Elements

 ANSYS displays the elements as solid elements, using a rectangular cross-section for link and shell elements and a circular cross-section for pipe elements. The crosssection proportions are determined from the real constant values.

FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

Sections

٢ ANSYS Main Menu Preferences Preprocessor Real Constants Material Props Sections Section Library Beam Shell Pretension List Sections Delete Section Meshing Checking Ctrls **H** Numbering Ctrls Archive Model Coupling / Cegn FLOTRAN Set Up FSI Set Up **⊞** Loads **B** Physics Path Operations **E** Solution General Postproc ⊞ TimeHist Postpro
 Topological Opt **E ROM Tool** Design Opt Run-Time Stats Session Editor 🔤 Finish

Building a model using <u>BEAM44</u>, <u>BEAM188</u>, or <u>BEAM189</u>, you can use the section commands (<u>SECTYPE</u>, <u>SECDATA</u>, etc.) or their GUI path equivalents to define and use cross sections in your models.

Sections

- A cross section defines the geometry of the beam in a plane perpendicular to the beam axial direction. ANSYS supplies a library of eleven commonly-used beam cross section shapes, and permits user-defined cross section shapes.
- When a cross section is defined, ANSYS builds a numeric model using a nine node cell for determining the properties (Iyy, Izz, etc.) of the section and for the solution to the Poisson's equation for torsional behaviour.

FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

Sections



FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

Geometry/Modelling

- Creating a solid model within ANSYS.
- Using direct generation.
- Importing a model created in a computeraided design (CAD) system.

Coordinate systems

- *Global* and *local* coordinate systems are used to locate geometry items (nodes, keypoints, etc.) in space.
- The *display* coordinate system determines the system in which geometry items are listed or displayed.
- The *nodal* coordinate system defines the degree of freedom directions at each node and the orientation of nodal results data.
- The *element* coordinate system determines the orientation of material properties and element results data.
- The *results* coordinate system is used to transform nodal or element results data to a particular coordinate system for listings, displays, or general postprocessing operations (POST1).
- The working plane, which is separate from the coordinate systems discussed in this chapter, is used to locate geometric primitives during the modeling process.

Coordinate systems

- (a) Cartesian (X, Y, Z components) coordinate system 0 (C.S.0)
- (b) Cylindrical (R, θ, Z components) coordinate system 1 (C.S.1)
- (c) Spherical (R, θ, φ components) coordinate system 2 (C.S.2)
- *(d) Cylindrical* (R, θ, Y components) coordinate system 5 (C.S.5)

Modeling (coordinates)



FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

Geometry/Modelling



24

Modeling - Create





- The hierarchy of modeling entities is as listed below:
 - Elements (and Element Loads)
 - Nodes (and Nodal Loads)
 - Volumes (and Solid-Model Body Loads)
 - Areas (and Solid-Model Surface Loads)
 - Lines (and Solid-Model Line Loads)
 - Keypoints (and Solid-Model Point Loads)

Geometric modeling

Computational Mechanics, AAU, Esbjerg

Modeling - Operate



Perform geometrical operations in order to obtain new geometrical entities

FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

Modeling - Move/Modify

ANSYS Main Menu ۲ Preferences Preprocessor Real Constants Material Props Modeling Create Operate Move / Modify ➢ Lines I Areas ➢ Volumes. **F** Nodes ⊞ Rotate Node CS
 F Elements ⊞ Transfer Coord ⊞ Reverse Normals
 ■ E Copy Reflect ⊕ Check Geom
 ■
 Delete Cyclic Sector 🧱 Genl plane strn 🧱 Update Geom ⊞ Checking Ctrls
 ⊞ Numbering Ctrls
 ⊞ Coupling / Cegn Loads -4 .

Move or modify locations or sizes of geometrical entities

FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

Modeling - Copy



Copy geometrical entities to new geometrical entities with new locations

FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

Modeling - Delete



- The hierarchy of modeling entities is as listed below:
 - Elements (and Element Loads)
 - Nodes (and Nodal Loads)
 - Volumes (and Solid-Model Body Loads)
 - Areas (and Solid-Model Surface Loads)
 - Lines (and Solid-Model Line Loads)
 - Keypoints (and Solid-Model Point Loads)

Computational Mechanics, AAU, Esbjerg

Modeling - Update Geom

NUpdate nodes using results file displa [UPGEOM] Update nodal coordinates using results file no	ocements odal displacements	×
FACTOR Scaling factor	1	
SBSTEP Load step		
Filename, Extension, Directory		Browse
OK Apply	Cancel	Help

Adds displacements from a previous analysis and updates the geometry of the finite element model to the deformed configuration.

Create – Keypoints (In Active CS)



Computational Mechanics, AAU, Esbjerg

Create – Lines (Straight Line)



FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

Create – Lines - Arcs

ANSYS Main Menu	\otimes
ferences	
processor	
lement Type	
.eal Constants	
1aterial Props	
ections	
1odeling	
] Create	
🗉 Keypoints	
🗆 Lines	
🗆 Arcs	
🏹 Through 3 KPs	
🏸 By End KPs & Rad	
冽 By Cent & Radius	
🏹 Full Circle	
🔊 Line Fillet	
E Areas	
⊞ Nodes	
Elements	
🔜 Contact Pair	
Diping Models	
🗉 Circuit	
🔤 Racetrack Coil	
Transducers	
3 Operate	
3 Move / Modify	
3 Copy	
3 Reflect	Ţ
] Chack Goom	Ľ

Arc by Center & Ra		
• Pick C Unpick		
Count = 0		
Maximum = 2		
Minimum = 2		
WP X =		
Y =		
Global X =		
Y =		
Z =		
C WP Coordinates		
🙃 Global Cartesian		
OK Apply		
Reset Cancel		
Help		

Geometric modeling

FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

Create – Areas (By 2 Corners)

ANSYS Main Menu (۲
ferences	
processor	
lement Type	
eal Constants	
1aterial Props	
ections	
1odeling	
] Create	
⊞ Keypoints	
⊞ Lines	
Areas	
Arbitrary	
🗆 Rectangle	
冽 By 2 Corners	
冽 By Centr & Cornr	
By Dimensions	
🗉 Circle	
Delygon	
🖉 Area Fillet	
Volumes	
⊡ Nodes	
🔤 Contact Pair	
Diping Models	
🗉 Circuit	
🔤 Racetrack Coil	
Transducers	
3 Operate	
3 Move / Modify	
3 Copy	
3 Reflect	
1 Check Geom	

\Lambda Rectangle by 2 C 🗵		
• Pick	C Unpick	
WPX =		
¥ =		
Global X =		
¥ =		
Z =		
WP X		
WP Y		
Width		
Height		
ок	Apply	
Reset	Cancel	
Help		

Create – Areas (By dimensions)



∧Create Recta	ngle by Dimensi	ions	×
[RECTNG] Create Red	tangle by Dimensions		
X1,X2 X-coordinates			
Y1,Y2 Y-coordinates			
ОК	Apply	Cancel	Help

Create – Areas (By Lines)

ANSYS Main Menu 🛞) 🛛 ANSYS Main Menu 🏾 🛞	9
eferences	• eferences	•
eprocessor	eprocessor	
Element Type	Element Type	
Real Constants	Real Constants	
Material Props	Material Props	
Sections	Sections	
Modeling	Modeling	
🗆 Create	🗆 Create	
🗉 Keypoints	🗄 Keypoints	
🗄 Lines	⊞ Lines	
🗉 Areas	🗆 Areas	
Arbitrary	Arbitrary	
🏹 Through KPs	n Through KPs	
🏹 Overlaid on Area	🖉 Overlaid on Area	
🖉 By Lines	🔁 By Lines	
🎘 By Skinning	🖉 By Skinning	
🖓 By Offset 🛛 🗧	🚽 🖉 By Offset 👘	
🗄 Rectangle	🗉 Rectangle	
🗉 Circle	🗉 Circle	
Polygon	Polygon	
🔊 Area Fillet	🖉 Area Fillet	
⊞ Nodes		
🗄 Elements		
🔟 Contact Pair	🔤 Contact Pair	
	Piping Models	
🖽 Circuit	🕀 Circuit	
🔤 Racetrack Coil	🔤 Racetrack Coil	
⊞ Transducers	Transducers	
🗄 Operate	🕀 Operate	
Move / Modify	■ Move / Modify	-1
•		

Create Are	ea by Li	
• Pick	C Unpick	
🖲 Single	C Box	
C Polygon	\mathbf{C} Circle	
C Loop		
Count =	0	
Maximum =	4	
Minimum =	2	
Line No. =		
O List of Items O Min. Max. Inc.		
OK	Apply	
Reset	Cancel	
Reset Pick All	Cancel Help	

FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

Create - Volumes



Computational Mechanics, AAU, Esbjerg

Booleans - Intersect

LINL (Line Intersect Line)



AINA (Area Intersect Area)



FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

Booleans - Intersect

Geometric modeling

VINV (Volume Intersect Volume)

LINA (Line Intersect Area)



New volume.



New keypoint



same surface)

(L1, A1 lie on the

FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

 $\sqrt{2}$

V1

39

Booleans - Intersect

LINV (Line Intersect Volume)

AINV (Area Intersect Volume)





Booleans - Add

AADD (Add Areas)

VADD (Add Volumes)







One complex volume (no internal boundaries)

LSBL (Line Subtract Line)

ASBA (Area Subtract Area)



FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

VSBV (Volume Subtract Volume)

LSBA (Line Subtract Area)



FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

LSBV (Line Subtract Volume)

ASBV (Area Subtract Volume)





FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

ASBL (Area Subtract Line)

VSBA (Volume Subtract Area)



FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

Booleans - Overlap

LOVLAP (Line Overlap Line)

AOVLAP (Area Overlap Area)



VOVLAP (Volume Overlap Volume)



FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg



VGLUE (Volume Glue Volume)



Geometric modeling

V3 and V4 share 4 keypoints, 4 lines, and an area

FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg 47

Mesh Generation Approaches

Structured discretization Mapped meshing

Unstructured discretization Free meshing



Mesh Attributes



▲Line Attributes	×
[LATT] Assign Attributes to Picked Lines	
MAT Material number	
REAL Real constant set number	1 💌
TYPE Element type number	1 BEAM3 💌
SECT Element section	None defined 👻
Pick Orientation Keypoint(s)	□ No
OK Apply	Cancel Help

FEM – ANSYS Classic Geometric modeling Computational Mechanics, AAU, Esbjerg

Meshing – Size Cntrls



Geometric modeling

Computational Mechanics, AAU, Esbjerg

FEM – ANSYS Classic

Meshing - ManualSize



FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

Meshing - Lines



FEM – ANSYS Classic Computational Mechanics, AAU, Esbjerg

Meshing - Clear



Deletes nodes and area elements associated with selected lines, areas, or volumes.