Course in
FEM – ANSYS Workbench/CAD
3D Solids
FEM - ANSYS Classic

• Lecture 1 - Introduction:
  – Introduction to FEM
  – ANSYS Basics
  – Analysis phases
  – Geometric modeling
  – The first model: Beam model
• Lecture 2 - Preprocessor:
  – Geometric modeling
  – Specification of Element type, Real Constants, Material, Mesh
  – Frame systems
  – Truss systems
  – Element tables
• Lecture 3 - Loads:
  – Boundary conditions/constraints/supports
  – Loads
  – Mesh attributes, meshing
  – Sections
• Lecture 4 – 2D plane models:
  – 2D Plane Solid systems
  – Geometric modeling
  – Postprocessing
• Lecture 5 – Analysis types:
  – Analysis types
  – Modal analysis
  – Buckling analysis
FEM - ANSYS Workbench/CAD

- Lecture 6 – 3D Solids:
  - 3D solid models
  - Booleans
  - Meshing issues
- Lecture 7 – 3D Modeling:
  - Operate
  - Import CAD
  - Advanced topics
- Lecture 8 – Analysis types:
  - Analysis types
  - Postprocessing
  - TimeHistProc
- Lecture 9 – Workbench basics:
  - Workbench basics
  - Geometric modeling
- Lecture 10 – Workbench analysis:
  - Workbench analysis types
Citation of the day

“Finite Element Analysis makes a good engineer great, and a bad engineer dangerous!”

Robert D. Cook,
Professor of Mechanical Engineering,
University of Wisconsin, Madison
Modeling considerations

• 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.
Modeling considerations

• 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
Modeling considerations

Real model
Continuum

Each point have an infinite number of deformation state variables, i.e. degrees of freedom (dof)

Transformation

Each point have a finite number of deformation state variables (u,v), i.e. degrees of freedom

Analysis model
Discrete
Modeling considerations
Modeling considerations
Modeling considerations
Modeling considerations

WHEEL LOADER REALIZATION
- a model and an architecture that enables simulation in a process context

By Ulf Sellgren

ISRN KTH/MMK/R—03/07—SE
TRITA-MMK 2003:07
ISSN 1400-1179
Modeling considerations
Modeling considerations

Figure 12. An Ansys beam representation of a lifting unit and rear frame.
Modeling considerations

Name: Liftarm

Name: LiftarmFE
Level of abstraction: Detailed

Model size
26171 nodes
78513 DOFs
13602 parabolic tet's
2 linear beams
9 rigid beams

Mating feature

Name: LiftarmBeams
Level of abstraction: Abstracted

Model size
73 nodes
438 DOFs
72 linear beams

Mating feature
Modeling considerations

- Characterization of problem
Modeling considerations

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

• Creating a solid model within ANSYS.
• Using direct generation.
• Importing a model created in a computer-aided 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)

- **Cartesian**
  - $(x, y, z)$

- **Cylindrical**
  - $(r, \theta, z)$

- **Spherical**
  - $(r, \theta, \phi)$

**General Curvilinear Coordinates**

**General orthogonal Coordinates**
Booleans
Geometry/Modelling

- Create – geometrical entities
- Operate – perform Boolean operations
- Move / Modify – move or modify geometrical entities
- Copy – copy geometrical entities
- Delete – geometrical entities
- Update Geom – update the geometry in relation to for example buckling analysis
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)
Example - Grinding shield
Example - Grinding shield
Example - Grinding shield
Example - Grinding shield
Modeling considerations

- Model storage *.lgw or *.db?
- Element type?
- Level of detail?
- Mesh method?
- Allow model modifications?
- Type of analysis to perform?
- Material models?
- Boundary conditions and loads?
Example - Grinding shield
Modeling considerations

Make a directory, e.g. c:\fem\grinding-shield
Change directory in ANSYS to c:\fem\grinding-shield
Make a new text file in c:\fem\grinding-shield
Rename from “New textdocument.txt” to “grinding-shield-kp.lgw”
Open with, e.g. Notepad
Enter the text shown

• Model storage *.lgw or *.db?
• Element type?
• Level of detail?
• Mesh method?
• Allow model modifications?
• Type of analysis to perform?
• Material models?
• Boundary conditions and loads?
Example - Grinding shield
Modeling considerations

/PREP7
LSTR, 1, 2
LSTR, 3, 4
LSTR, 5, 6
LSTR, 6, 7
LSTR, 7, 8
LSTR, 8, 9
LSTR, 9, 10
LSTR, 11, 12
LSTR, 13, 14
LSTR, 14, 1
!*
L2TAN,1,2
!*  
L2TAN,-9,-8
!*  
L2TAN,2,3
!*  
L2TAN,-8,-7
LSTR, 9, 6
LSTR, 5, 10
LSTR, 4, 11
LSTR, 3, 12
LSTR, 2, 13

Make a new text file in c:\fem\grinding-shield
Rename from “New textdocument.txt” to “grinding-shield-lines.lgw”
Open with, e.g. Notepad
Enter the text shown

- Model storage *.lgw or *.db?
- Element type?
- Level of detail?
- Mesh method?
- Allow model modifications?
- Type of analysis to perform?
- Material models?
- Boundary conditions and loads?
Example - Grinding shield
Modeling considerations

Make a new text file in c:\fem\grinding-shield
Rename from “New textdocument.txt” to “grinding-shield-areas.lgw”
Open with, e.g. Notepad
Enter the text shown

A GOOD TIME TO SAVE AS *.db

- Model storage *.lgw or *.db?
- Element type?
- Level of detail?
- Mesh method?
- Allow model modifications?
- Type of analysis to perform?
- Material models?
- Boundary conditions and loads?
Modeling - Operate

Perform geometrical operations in order to obtain new geometrical entities
Example - Grinding shield

Modeling considerations

Enter the command sequence in the command line

\PREP7
VROTAT,1,2,3,4,5,6,8,7,180, ,

A GOOD TIME TO SAVE AS *.db

- Model storage *.lgw or *.db?
- Element type?
- Level of detail?
- Mesh method?
- Allow model modifications?
- Type of analysis to perform?
- Material models?
- Boundary conditions and loads?
Example - Grinding shield
Modeling considerations

Enter the command sequence in the command line

\PREP7
VOFFST,7,40,
VOFFST,8,40,
VOFFST,9,40,

A GOOD TIME TO SAVE AS *.db

• Model storage *.lgw or *.db?
• Element type?
• Level of detail?
• Mesh method?
• Allow model modifications?
• Type of analysis to perform?
• Material models?
• Boundary conditions and loads?
Modeling - Move/Modify

Move or modify locations or sizes of geometrical entities
Modeling - Copy

Copy geometrical entities to new geometrical entities with new locations
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)
Modeling - Update Geom

Add displacements from a previous analysis and updates the geometry of the finite element model to the deformed configuration.
Booleans - Intersect

LINL (Line Intersect Line)

AINA (Area Intersect Area)
Booleans - Intersect

**VINV (Volume Intersect Volume)**

**LINA (Line Intersect Area)**

- VINV: Intersect two volumes and create a new volume.
- LINA: Intersect a line with an area and create a new line.

**Examples:**
- VINV: Intersection of two volumes results in a new volume.
- LINA: Intersection of a line with an area results in a new line.
Booleans - Intersect

LINV (Line Intersect Volume)

L1

V1

New line

A1

V1

New area

AINV (Area Intersect Volume)
Booleans - Add

AADD (Add Areas)

VADD (Add Volumes)

One complex volume (no internal boundaries)
Booleans - Subtract

**LSBL (Line Subtract Line)**

L1 - L2 → L1 - L2

L1 and L2 are lines. L1 - L2 results in a new line.

**ASBA (Area Subtract Area)**

A1 - A2 → A1 - A2

A1 and A2 are areas. A1 - A2 results in a new area.
Booleans - Subtract

VSBV (Volume Subtract Volume)  
LSBA (Line Subtract Area)
Example - Grinding shield
Modeling considerations

Enter the command sequence in the command line

\PREP7
VSBV,ALL, 13
!*  
VSBV,ALL, 14
!*  
VSBV,ALL, 15

- Model storage *.lgw or *.db?
- Element type?
- **Level of detail**?
- Mesh method?
- **Allow model modifications**?
- Type of analysis to perform?
- Material models?
- Boundary conditions and loads?
Booleans - Subtract

LSBV (Line Subtract Volume)  ASBV (Area Subtract Volume)

Common point
2 new lines

New area
Booleans - Subtract

ASBL (Area Subtract Line)  

VSBA (Volume Subtract Area)

FEM – ANSYS Workbench/CAD  
Computational Mechanics, AAU, Esbjerg  
3D Solids
Booleans - Overlap

LOVLAP (Line Overlap Line)

AOVLAP (Area Overlap Area)

VOVLAP (Volume Overlap Volume)
Booleans - Glue

LGLUE (Line Glue Line)

AGLUE (Area Glue Area)

VGLUE (Volume Glue Volume)

V3 and V4 share 4 keypoints, 4 lines, and an area
Meshing issues
Modeling considerations

• Run through example240 and example241
• What element type should be selected for the grinding shield example?
• Mesh method?
  • Model storage *.lgw or *.db?
  • Element type?
  • Level of detail?
  • Mesh method?
  • Allow model modifications?
  • Type of analysis to perform?
  • Material models?
  • Boundary conditions and loads?
Modeling considerations

- Apply the boundary conditions and the loads indicated
- Solve
- Plot the deformed shape
Example - Fins
Example - Fins

/PREP7
ET,1,SOLID90
K, ,,,, 
K, ,15,, ,
K, ,15,7.5,,
K, ,,7.5,,
! Round bottom surface
K, ,6,, ,
K, ,6,-1,,
K, ,,3,,
K, ,-1,3,,
! Round top surface
K, ,,,1,
K, ,5,,1,
K, ,5,-1,1,
K, ,,2,1,
K, ,-1,2,1,
! Extra points
K, ,5,,2,
K, ,,2,2,
Example - Fins

LSTR, 1, 5
LSTR, 5, 2
LSTR, 2, 3
LSTR, 3, 4
LSTR, 4, 7
LSTR, 7, 1
LSTR, 5, 6
LSTR, 10, 11
LSTR, 12, 13
LSTR, 7, 8
LSTR, 10, 14
LSTR, 12, 15
LSTR, 10, 9
LSTR, 9, 12
LSTR, 9, 1

!* 
L2TAN,-10,7
!* 
L2TAN,-9,8
!* 
L2TAN,-11,2
!* 
L2TAN,5,12
Example - Fins

lesize,3,,1,,
lesize,4,,1,,

lesize,16,,2,,
lesize,17,,2,,

lesize,1,,2,,
lesize,6,,2,,

lesize,13,,2,,
lesize,14,,2,,

!
lesize,18,,1,,
lesize,19,,1,,
lesize,15,,1,,

! Areas
AL,2,3,4,5,16
AL,16,6,1
AL,17,14,13
AL,15,13,1,18
AL,16,17,18,19
AL,6,19,14,15
Example - Fins

VEXT, 1, 2, , 0, 0, -6, , , ,
VEXT, 3, , , 0, 0, 49, , , ,
VA, 2, 3, 4, 5, 6

ACCAT, 9, 10

MSHAPE, 0, 3d
MSHKEY, 1
VMESH, 1, 4,

VSYMM, X, ALL, , , 0, 0
VSYMM, Y, ALL, , , 0, 0

VGEN, 3, ALL, , , 30, , , 0
VGEN, 2, ALL, , ,, 15 , , , 0