Course in ANSYS

Modeling reviewed – Boolean's

Course Outline

Introduction

Lesson 1. Modeling reviewed – Boolean's

Lesson 2. Boolean's + meshing issues

Lesson 3. Operate + meshing issues

Lesson 4. Import + meshing issues

Lesson 5. Meshing – advanced topics

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

Programme for Lesson:

BUILD THE MODEL

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

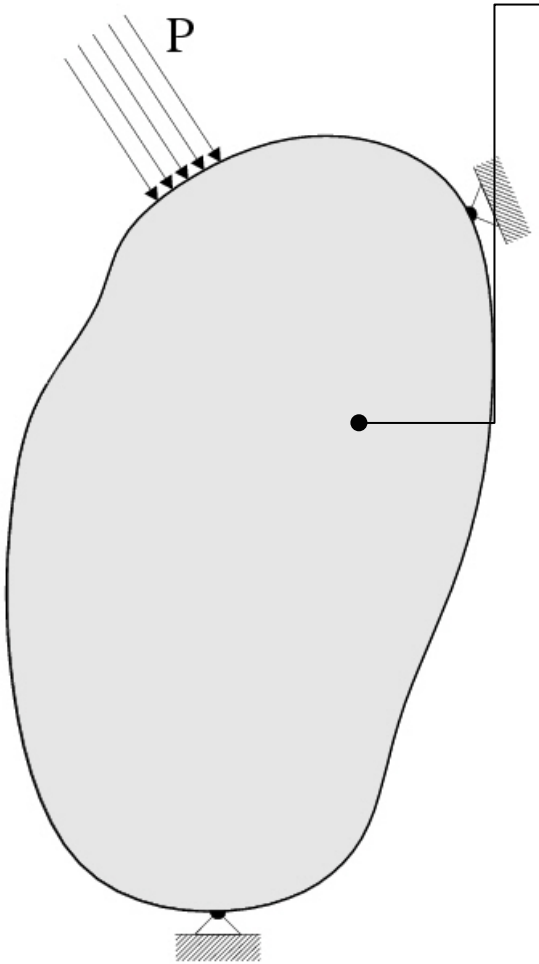
Modeling considerations

- 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

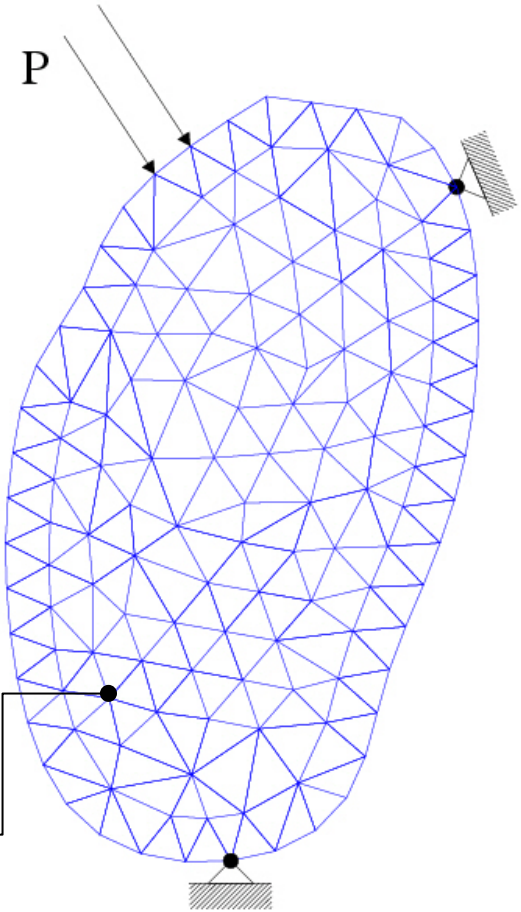


Real model
Continuum

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

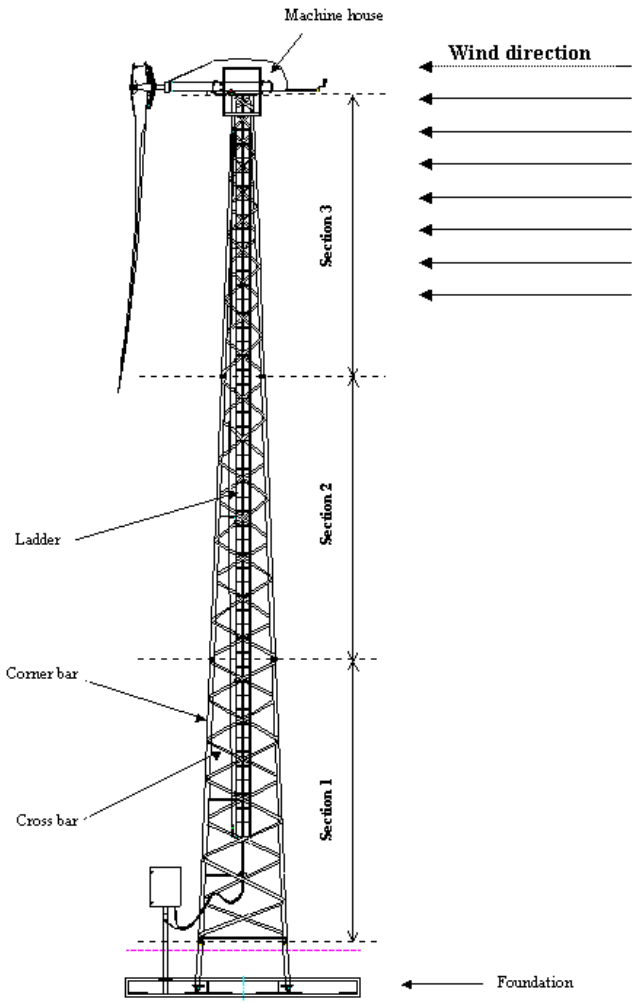
Transformation

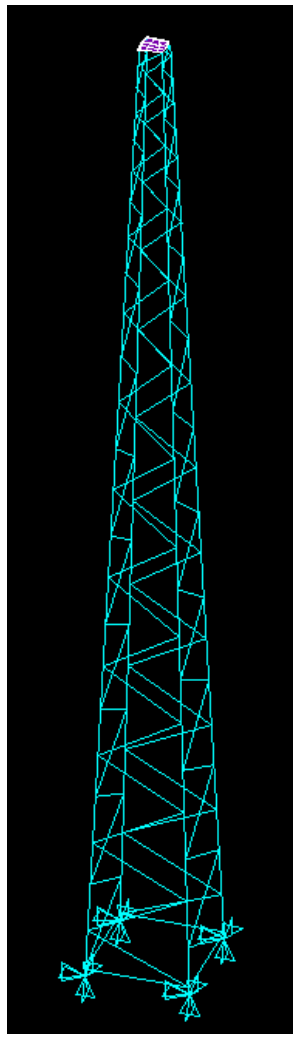
Each point have a **finite** number of deformation state variables (u,v), i.e. degrees of freedom



Analysis model
Discrete

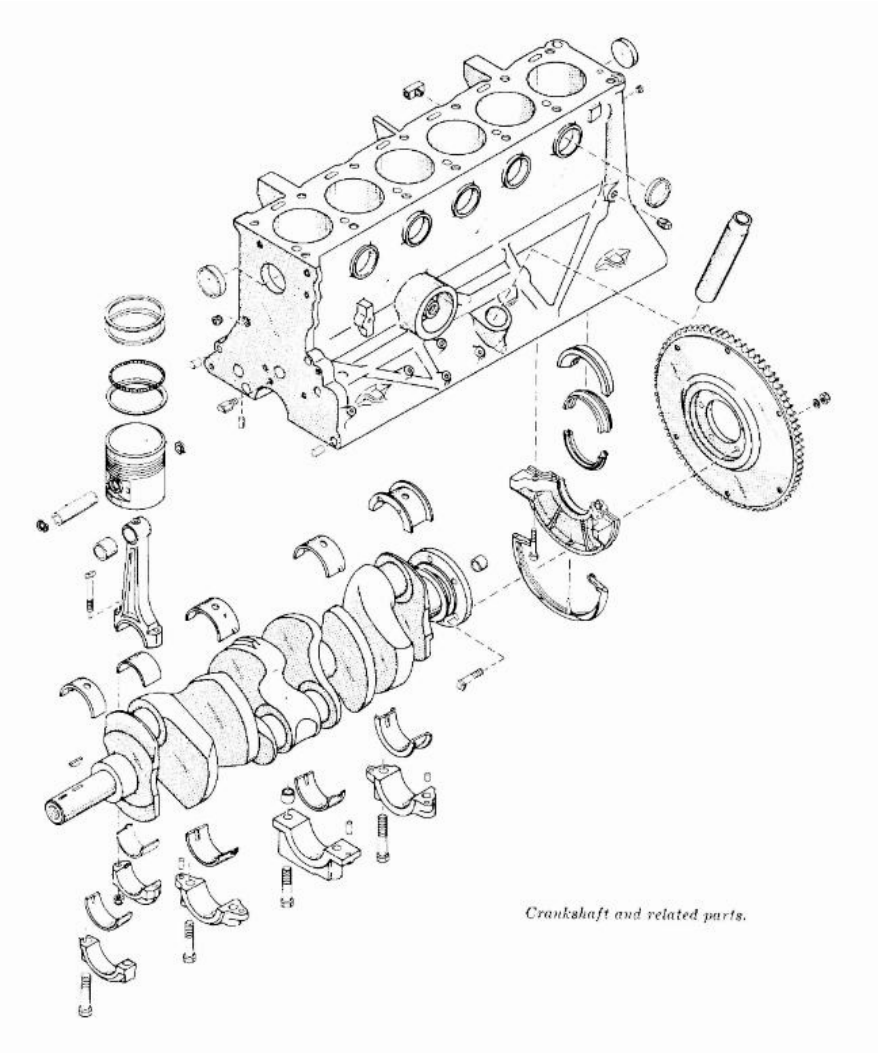
Modeling considerations





Modeling considerations

Modeling considerations



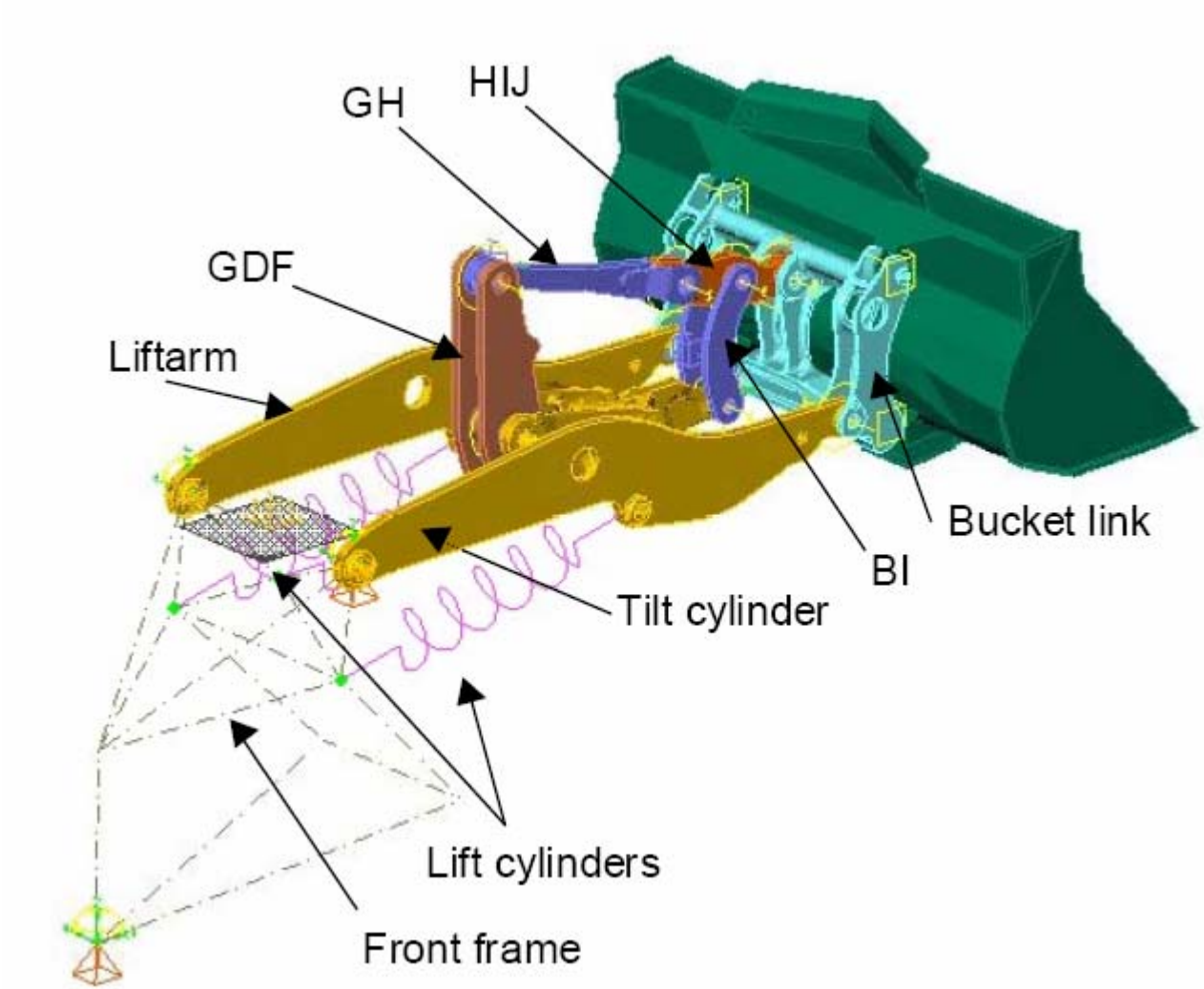
Modeling considerations

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

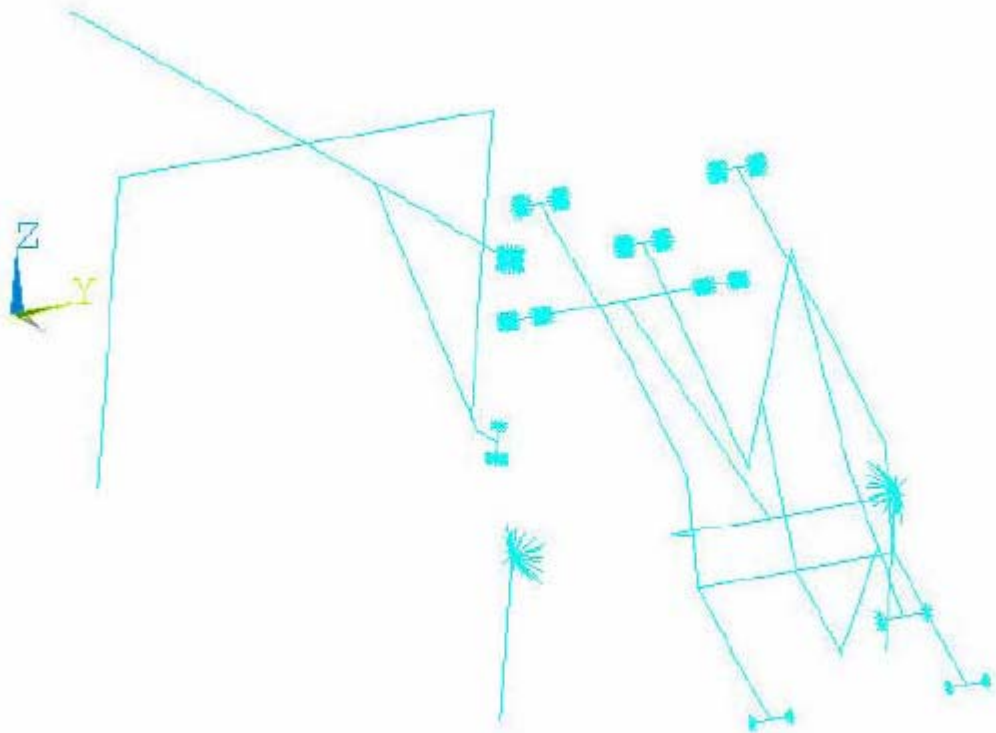
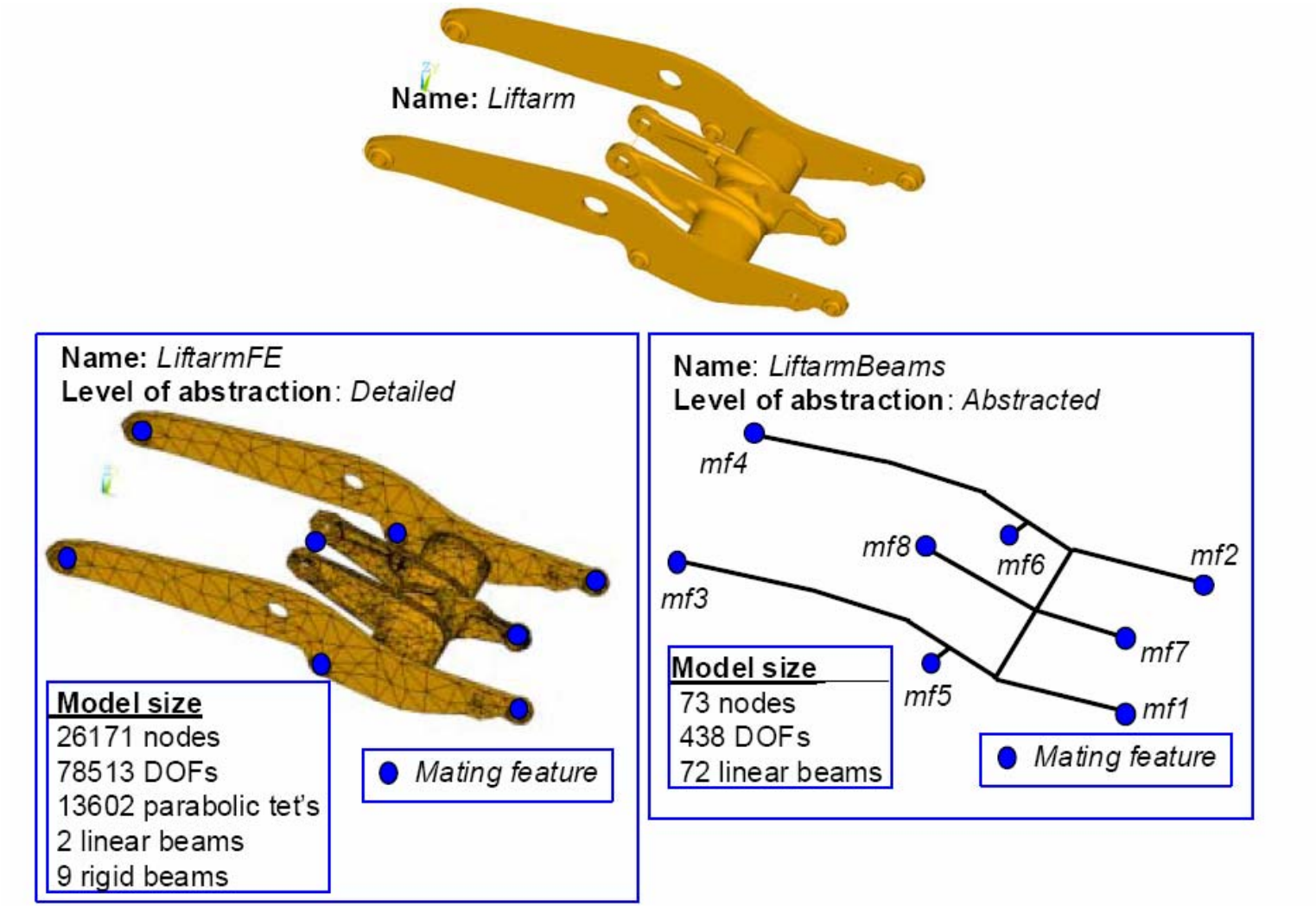


Figure 12. An Ansys beam representation of a lifting unit and rear frame.

Modeling considerations

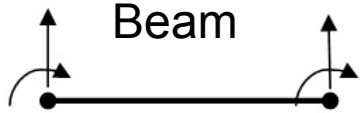


Modeling considerations

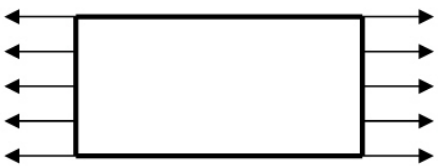
- Characterization of problem

Rod

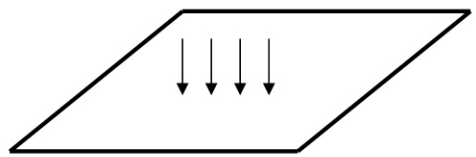
Beam



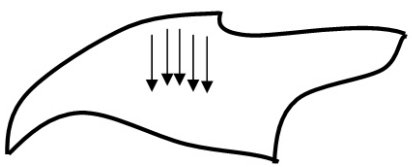
Disk



Plate



Shell



Solid

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.

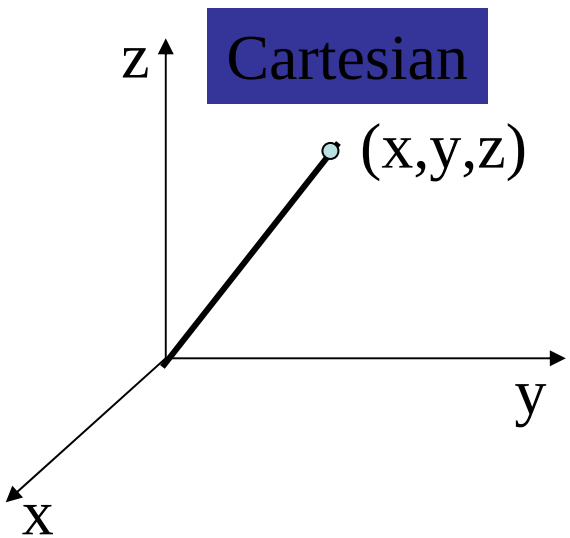
Coordinate systems

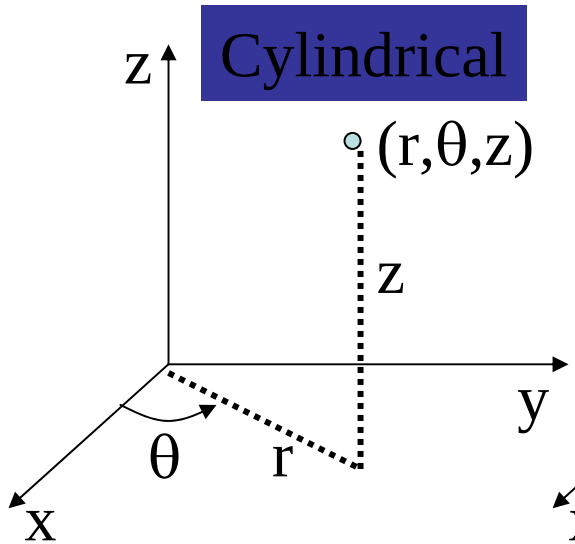
- *Global* and *local* coordinate systems are used to locate geometry items (nodes, keypoints, etc.) in space.
- The *display* coordinate system determines the system in which geometry items are listed or displayed.
- The *nodal* coordinate system defines the degree of freedom directions at each node 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.

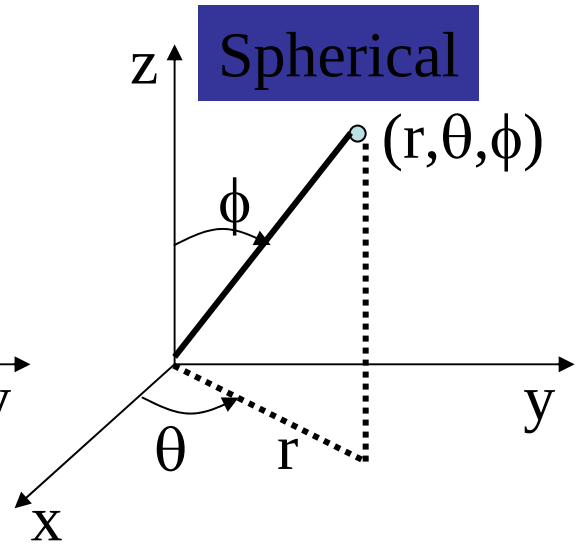
Coordinate systems

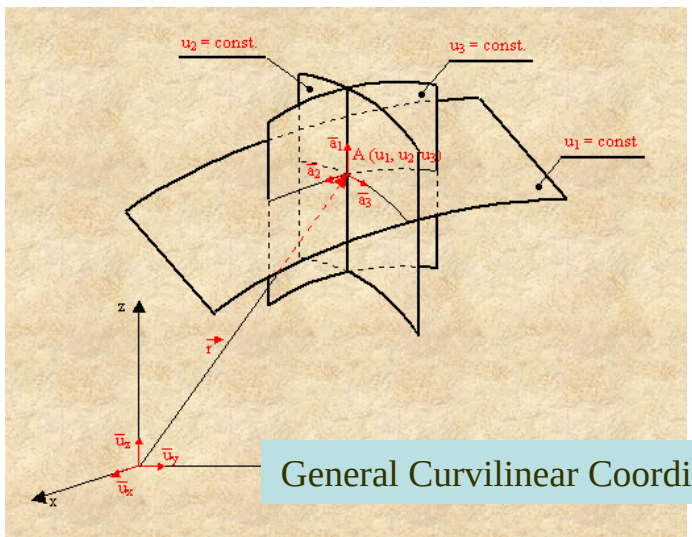
- *(a) Cartesian* (X, Y, Z components) coordinate system 0 (C.S.0)
- *(b) Cylindrical* (R, θ, Z components) coordinate system 1 (C.S.1)
- *(c) Spherical* (R, θ, φ components) coordinate system 2 (C.S.2)
- *(d) Cylindrical* (R, θ, Y components) coordinate system 5 (C.S.5)

Modeling (coordinates)

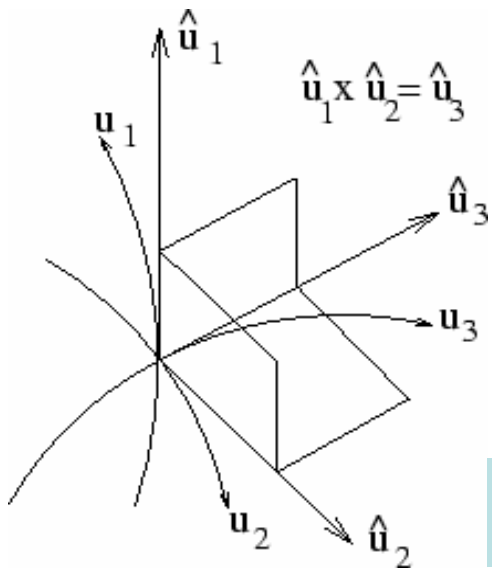






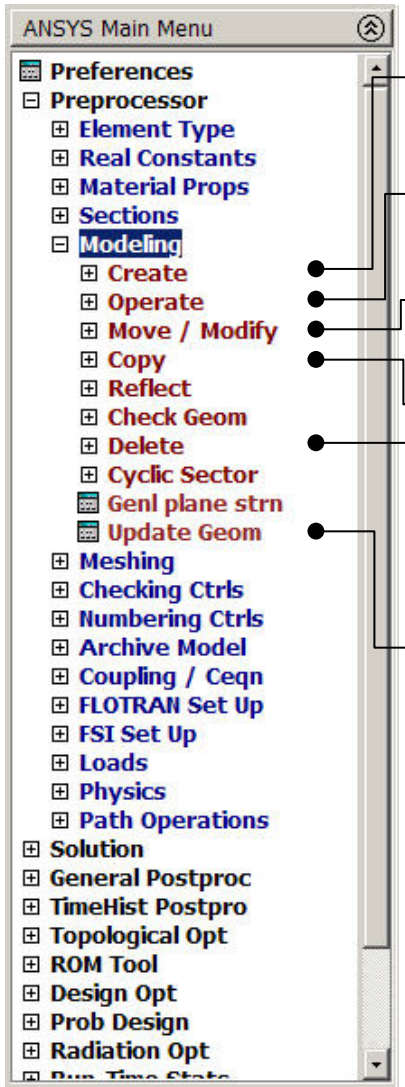


General Curvilinear Coordinates



General orthogonal Coordinates

Geometry/Modelling



Create – geometrical entities

Operate – perform Boolean operations

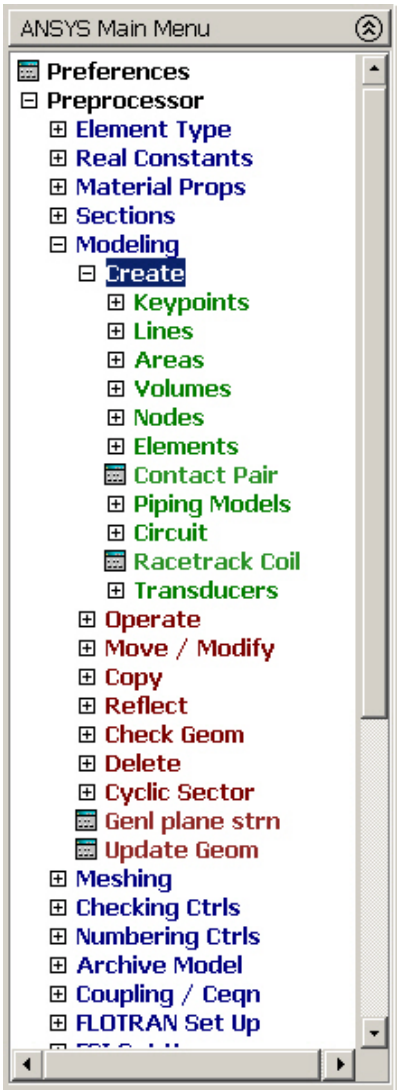
Move / Modify – move or modify geometrical entities

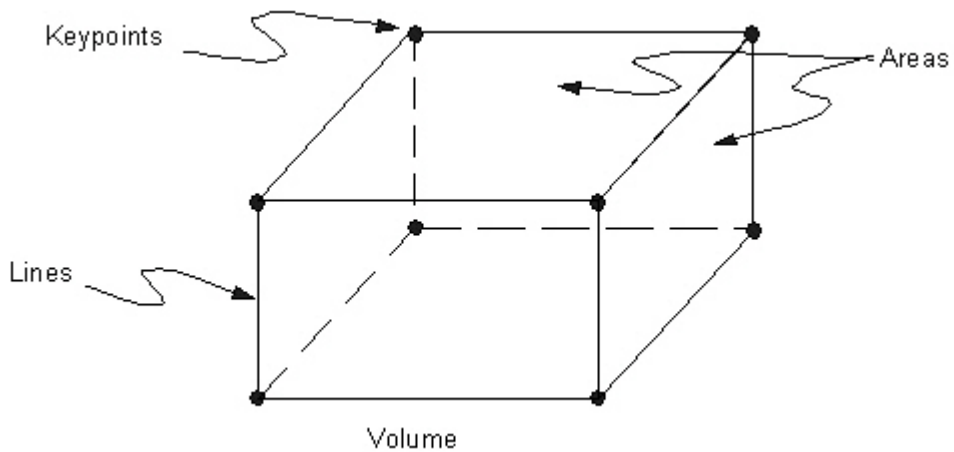
Copy – copy geometrical entities

Delete – geometrical entities

Update Geom – update the geometry in relation to for example buckling analysis

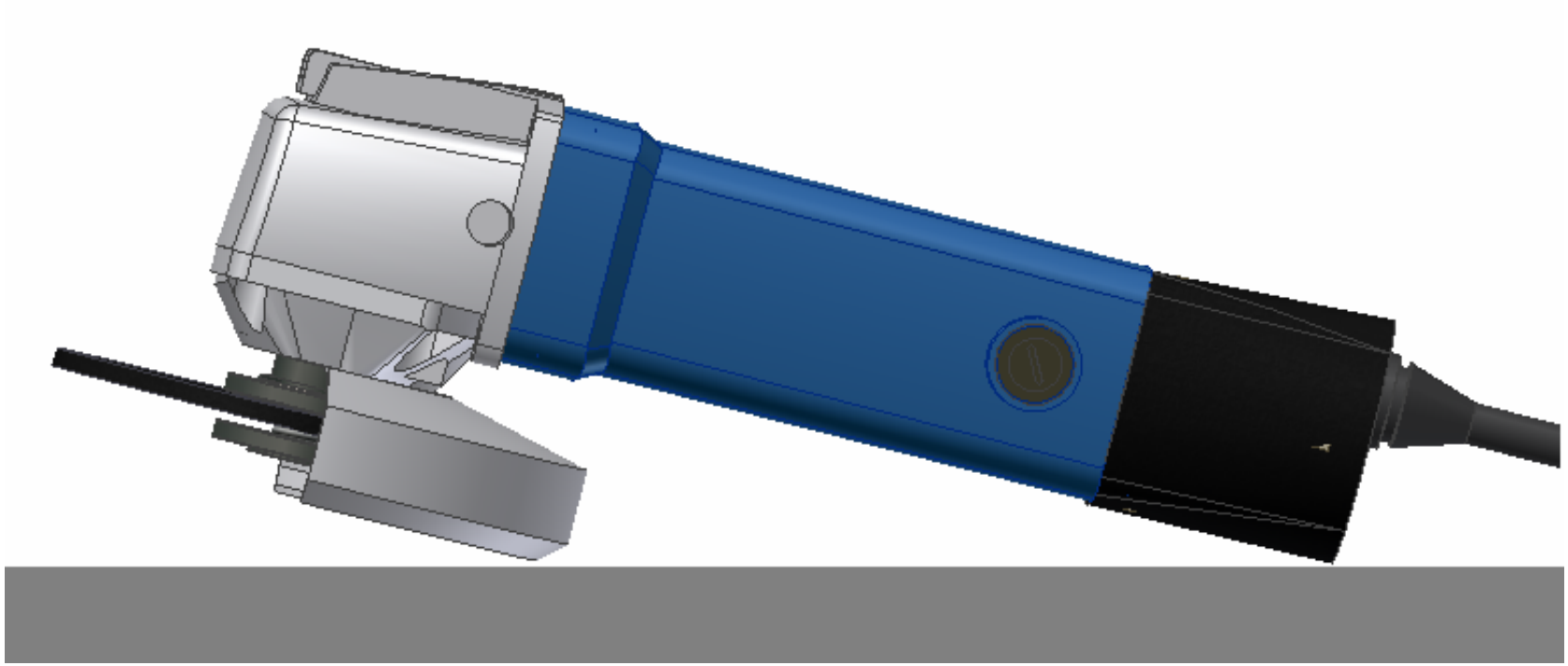
Modeling - Create



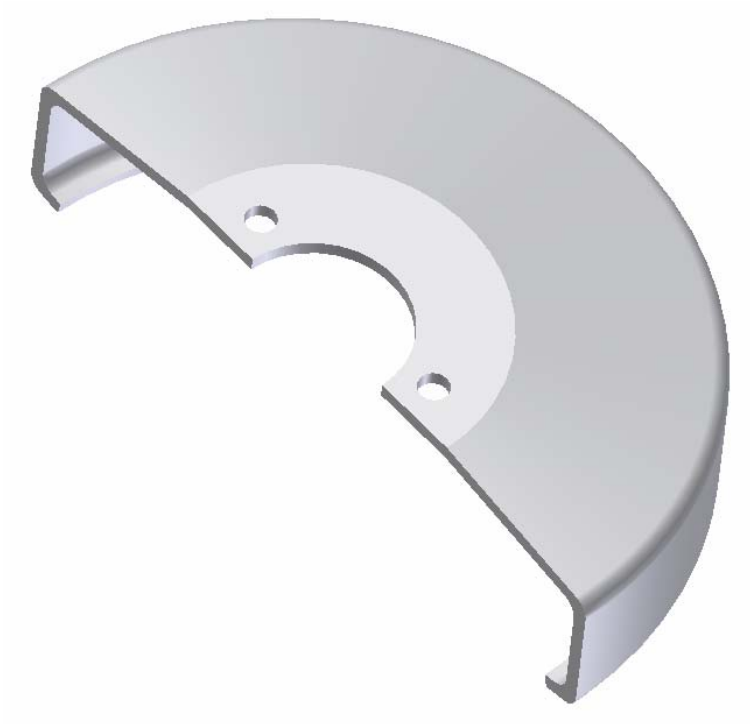


- The hierarchy of modeling entities is as listed below:
  - Elements (and Element Loads)
  - Nodes (and Nodal Loads)
  - Volumes (and Solid-Model Body Loads)
  - Areas (and Solid-Model Surface Loads)
  - Lines (and Solid-Model Line Loads)
  - Keypoints (and Solid-Model Point Loads)

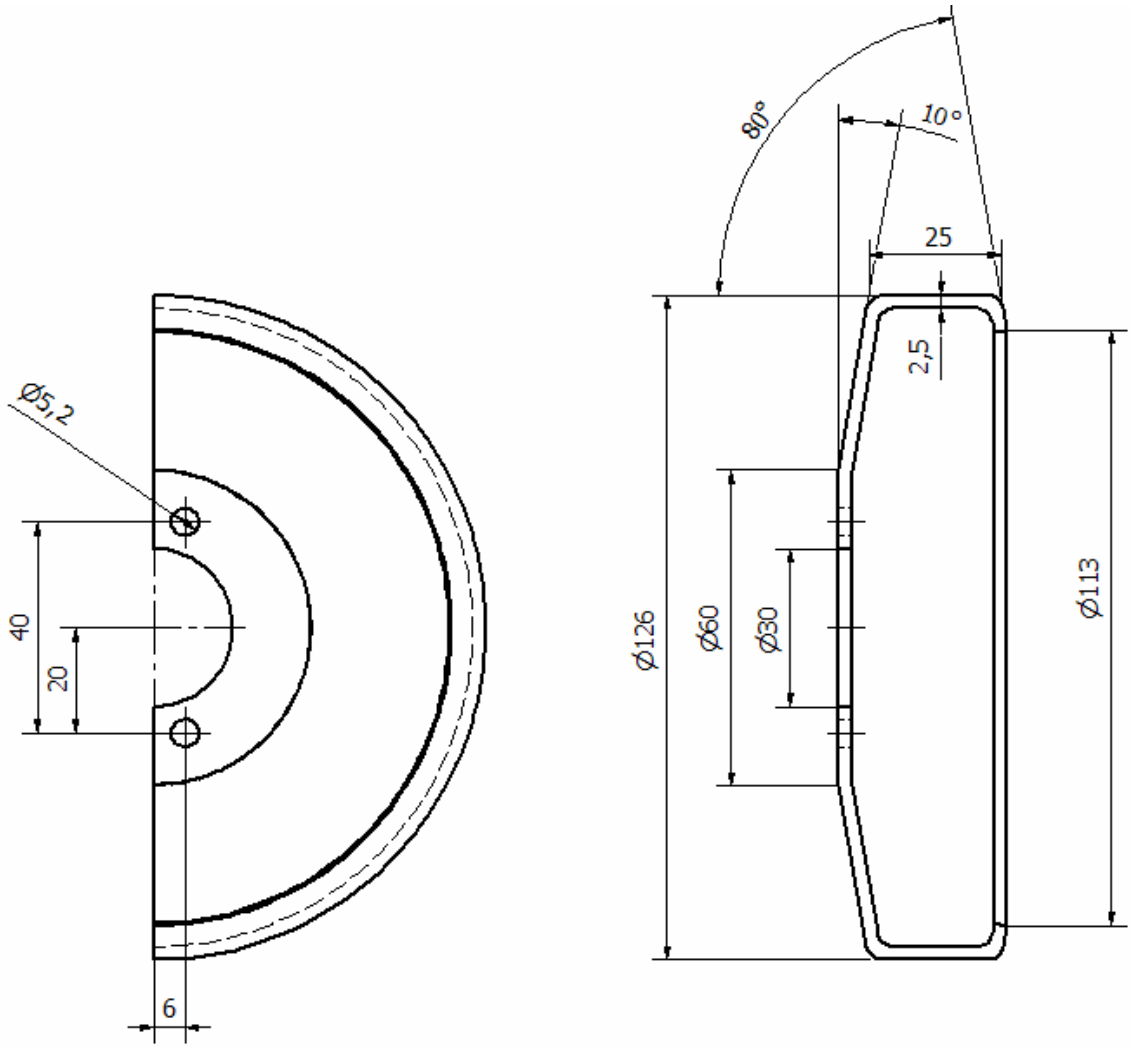
Example - Grinding shield



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

```
/PREP7
K,1,56,,,
K,2,60,,,
K,3,63,3,,
K,4,63,19,,
K,5,60,22,,
K,6,30,32,,
K,7,,32,,
K,8,,29.5,,
K,9,30,29.5,,
K,10,59.5,20,,
K,11,60.5,19,,
K,12,60.5,3,,
K,13,60,2.5,,
K,14,56,2.5,,
K,100,,,,
K,200,20,,-6,
K,201,20,40,-6,
K,300,-20,,-6,
K,301,-20,40,-6,
```

- 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

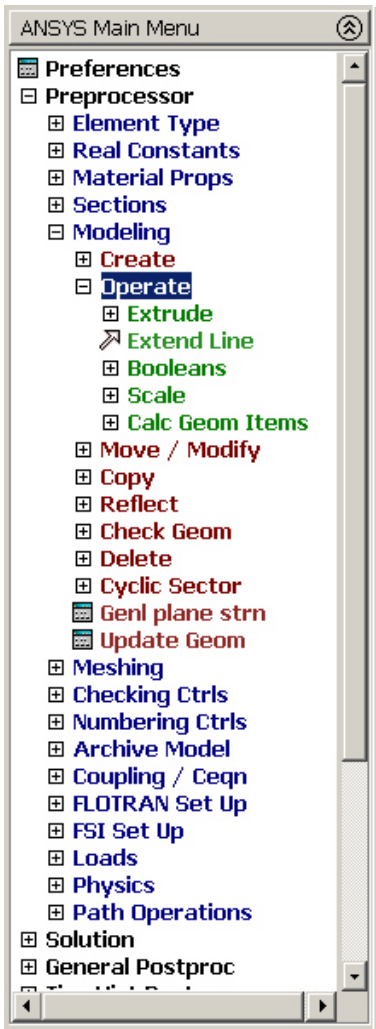
Make a new text file in c:\fem\grinding-shield
Rename from “New textdocument.txt” to “grinding-shield-areas.lgw”
Open with, e.g. Notepad
Enter the text shown

```
/PREP7
AL,4,5,6,15
AL,15,16,3,7
AL,16,13,17,14
AL,17,2,18,8
AL,18,12,19,11
AL,19,10,9,1
!*
CIRCLE,100,15,7, , ,
AL,20,21,22,23
!*
CIRCLE,200,3,201, , ,
AL,24,25,26,27
!*
CIRCLE,300,3,301, , ,
AL,28,29,30,31
```

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

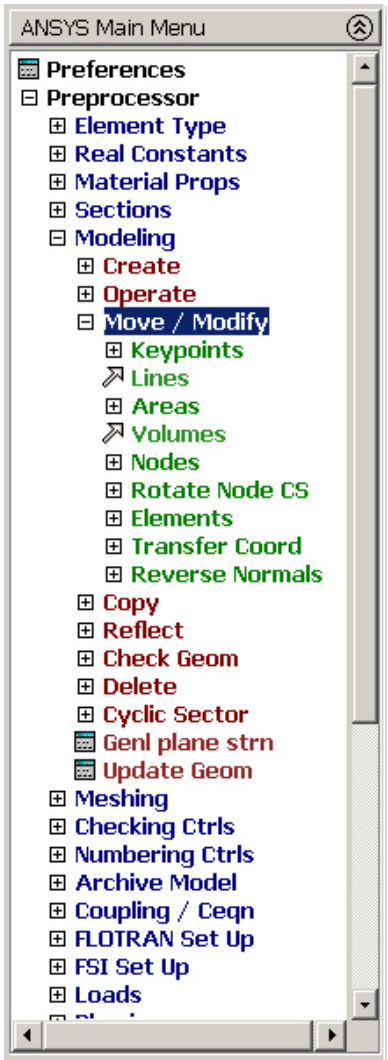
Enter the command sequence in the command line

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

A GOOD TIME TO SAVE AS *.db

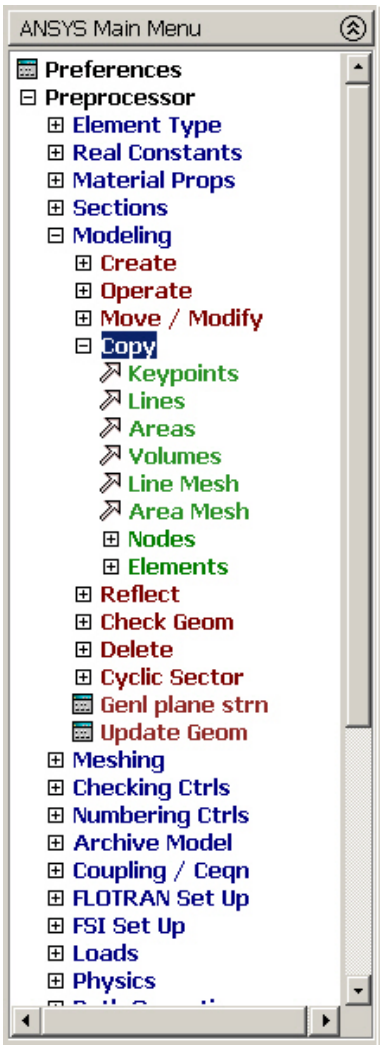
- Model storage ***.lgw** or *.db?
- Element type?
- **Level of detail?**
- Mesh method?
- **Allow model modifications?**
- **Type of analysis to perform?**
- Material models?
- Boundary conditions and loads?

Modeling - Move/Modify



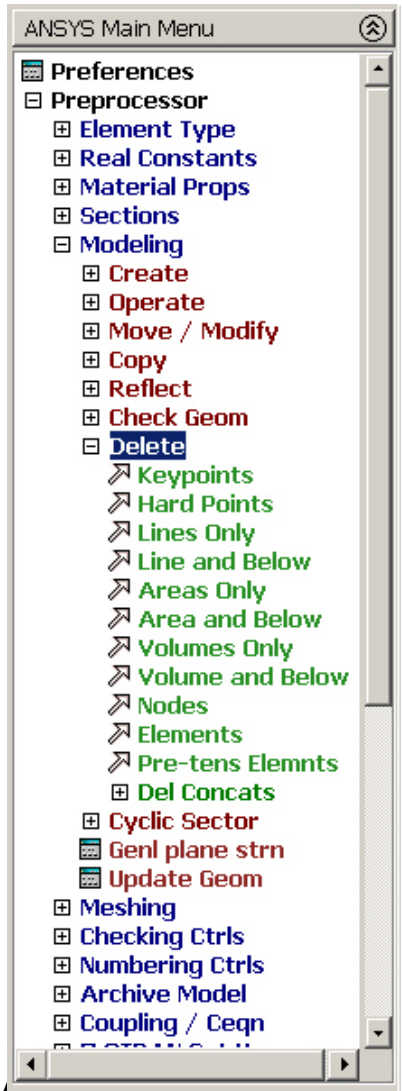
Move or modify locations or sizes of geometrical entities

Modeling - Copy



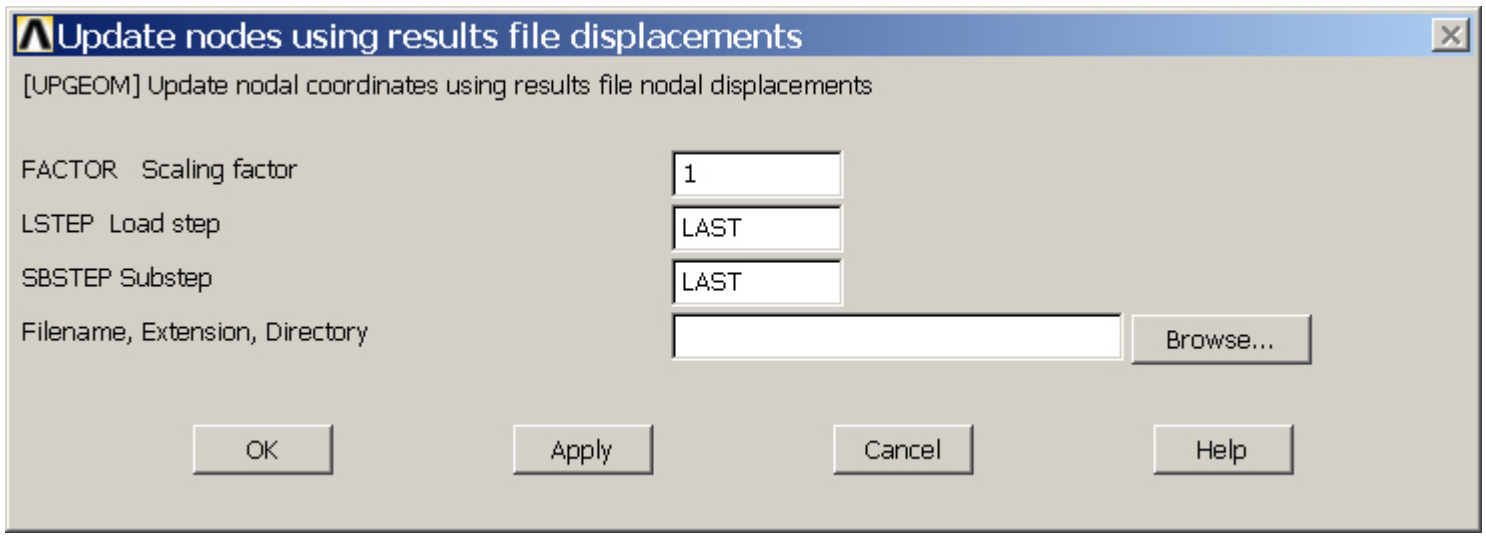
Copy geometrical entities to new geometrical entities with new locations

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)

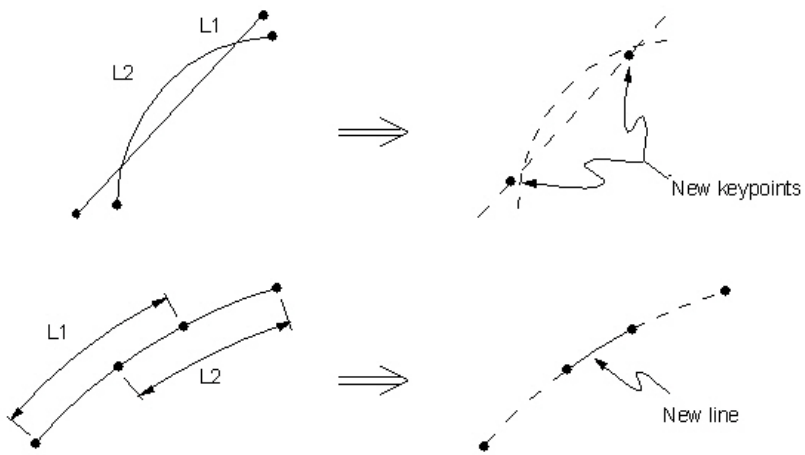
Modeling - Update Geom



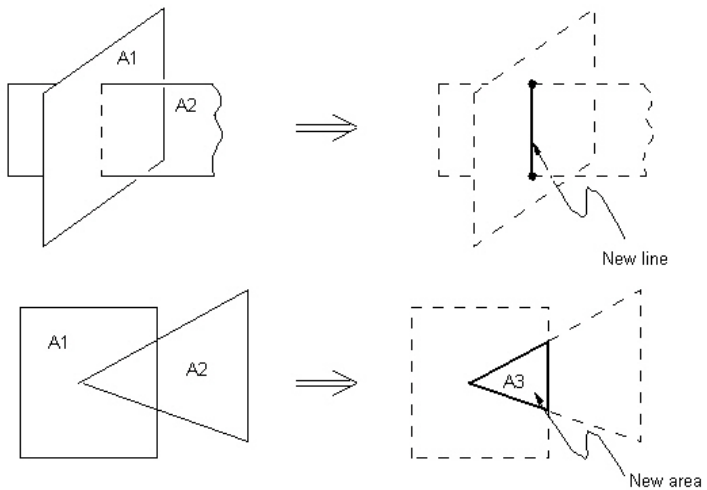
Adds displacements from a previous analysis and updates the geometry of the finite element model to the deformed configuration.

Booleans - Intersect

LINL (Line Intersect Line)

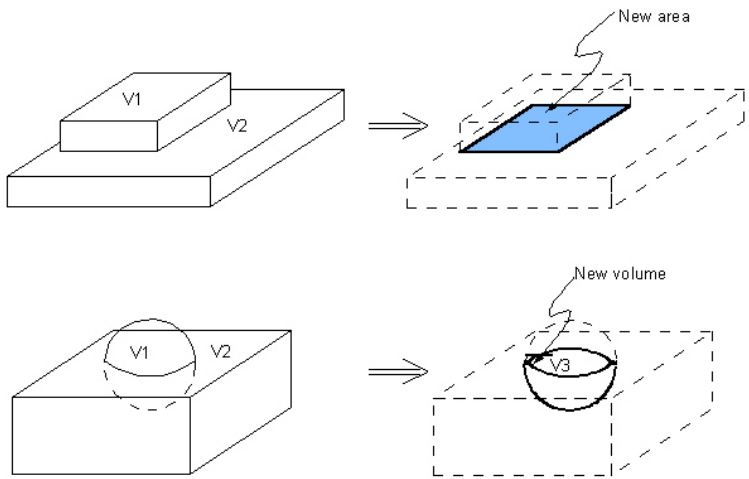


AINA (Area Intersect Area)

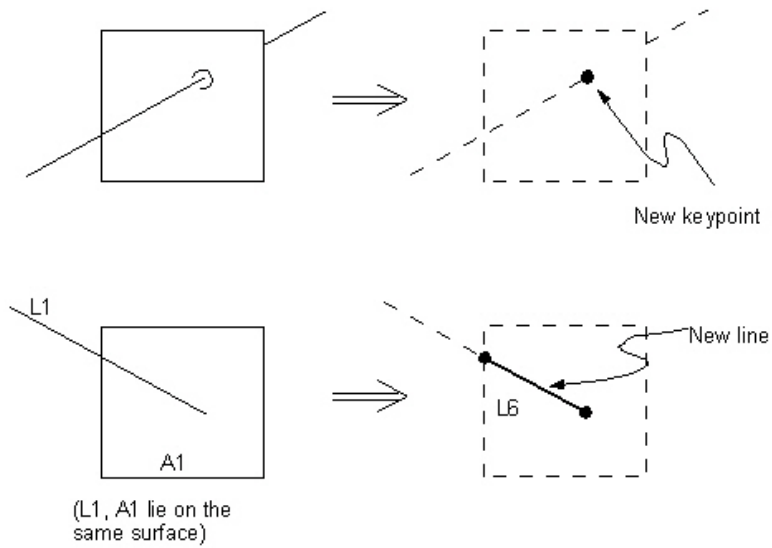


Booleans - Intersect

VINV (Volume Intersect Volume)

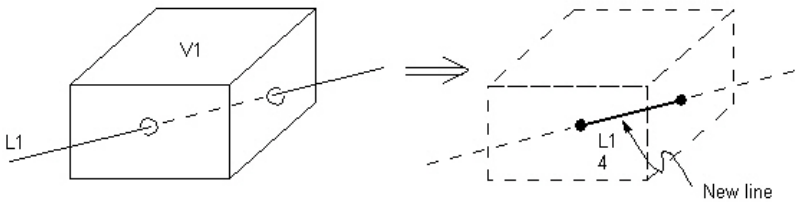


LINA (Line Intersect Area)

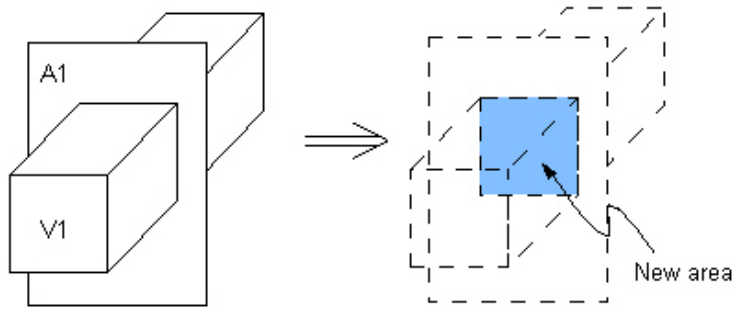


Booleans - Intersect

LINV (Line Intersect Volume)

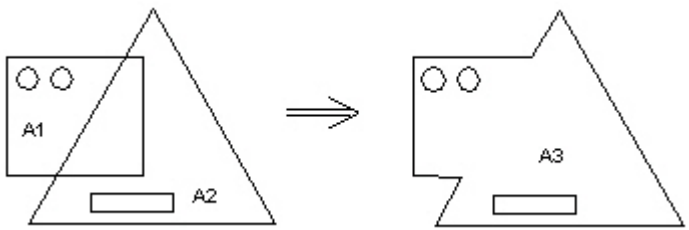


AINV (Area Intersect Volume)

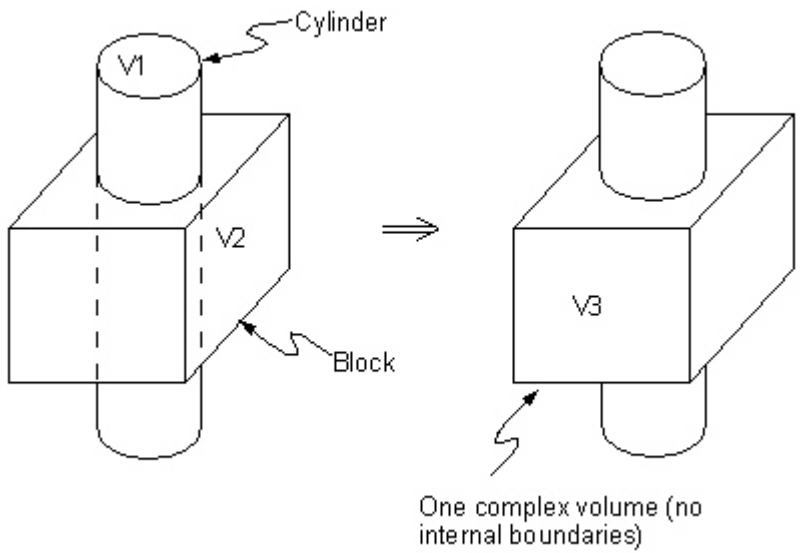


Booleans - Add

AADD (Add Areas)

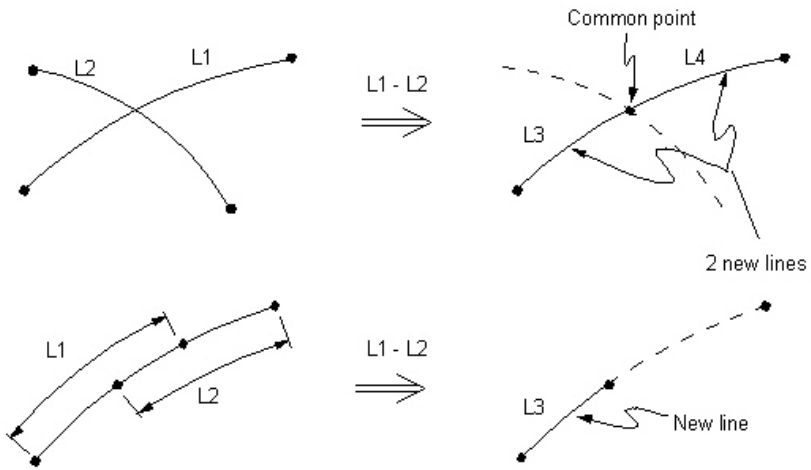


VADD (Add Volumes)

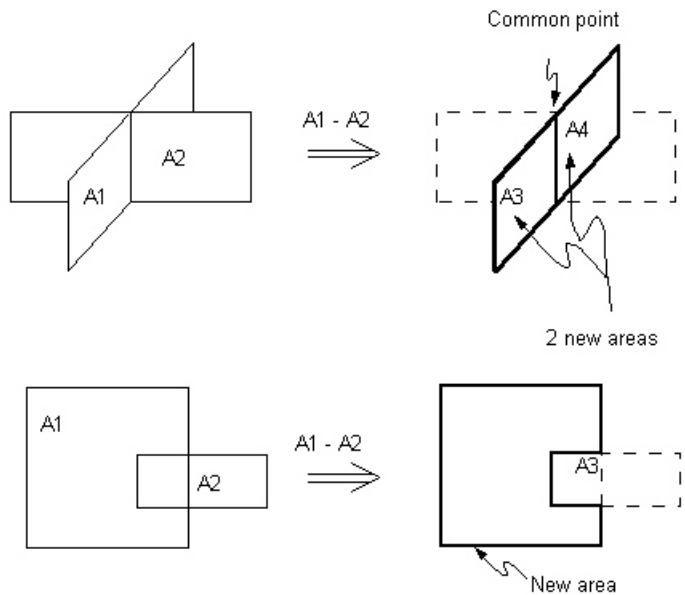


Booleans - Subtract

LSBL (Line Subtract Line)

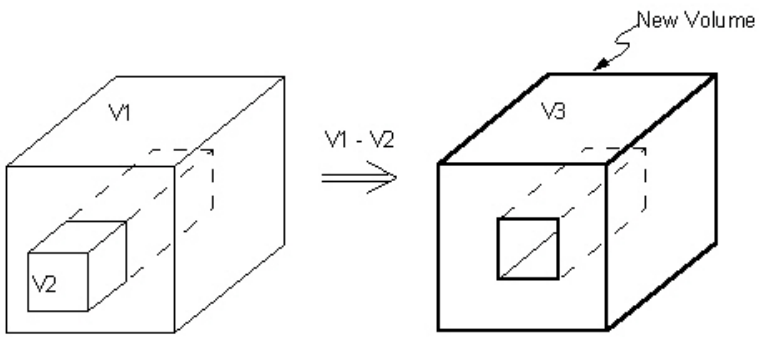


ASBA (Area Subtract Area)

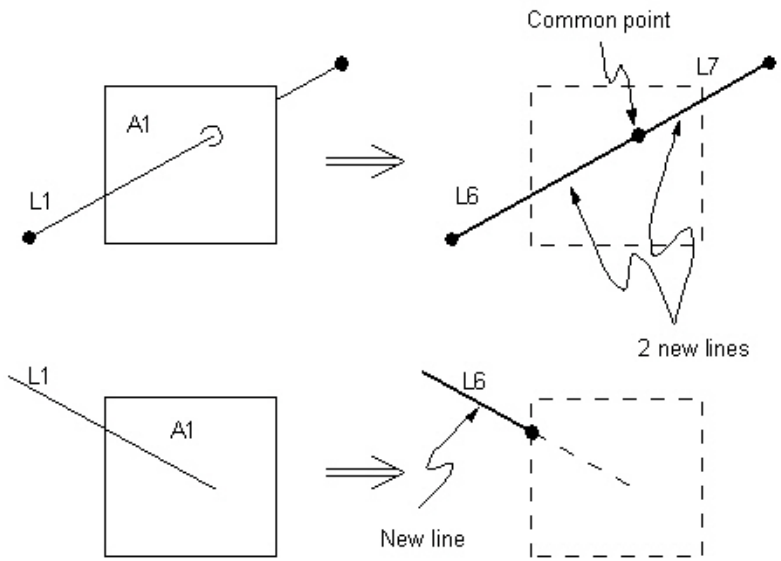


Booleans - Subtract

VSBV (Volume Subtract Volume)



LSBA (Line Subtract Area)



Example - Grinding shield Modeling considerations

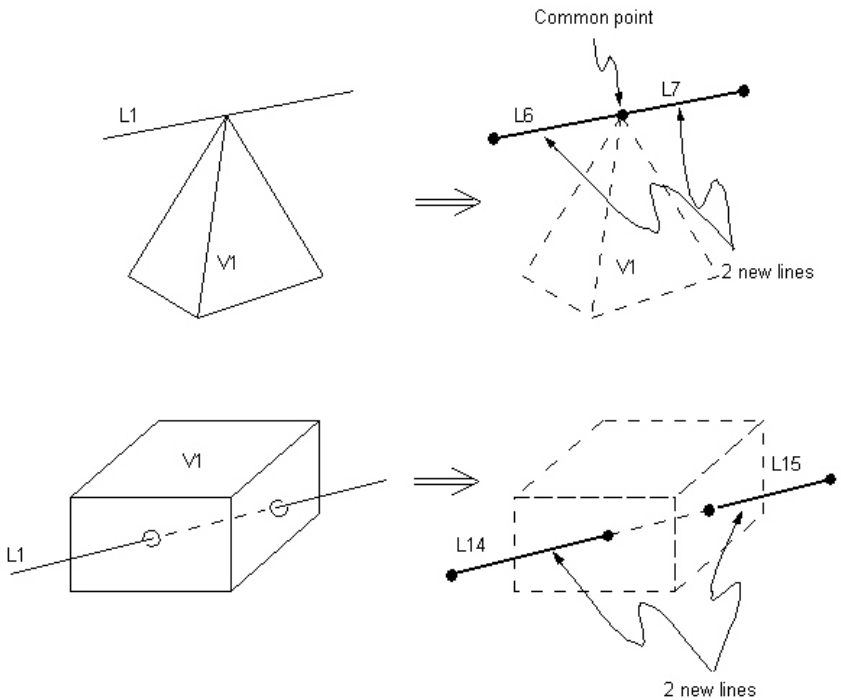
Enter the command sequence in the command line

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

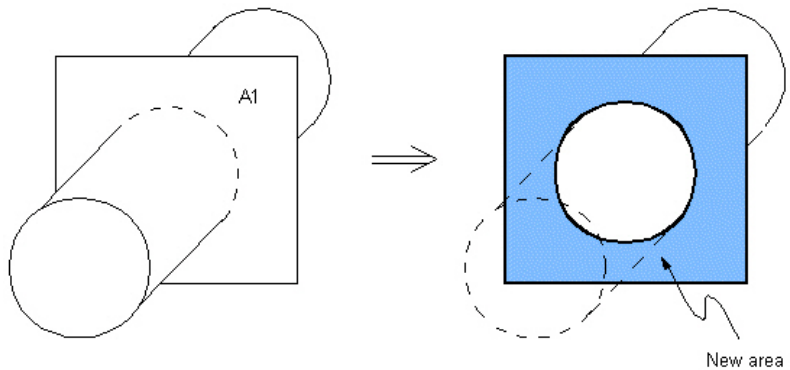
- Model storage ***.lgw** or *.db?
- Element type?
- **Level of detail?**
- Mesh method?
- **Allow model modifications?**
- **Type of analysis to perform?**
- Material models?
- Boundary conditions and loads?

Booleans - Subtract

LSBV (Line Subtract Volume)

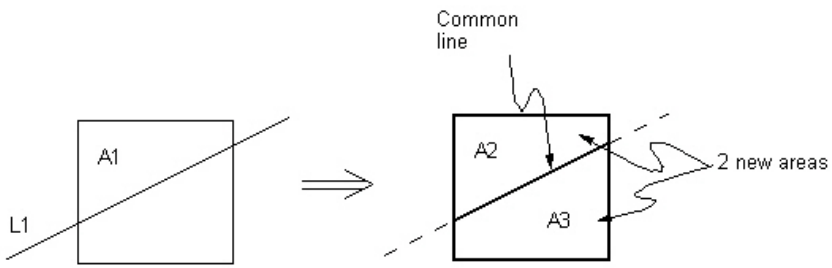


ASBV (Area Subtract Volume)

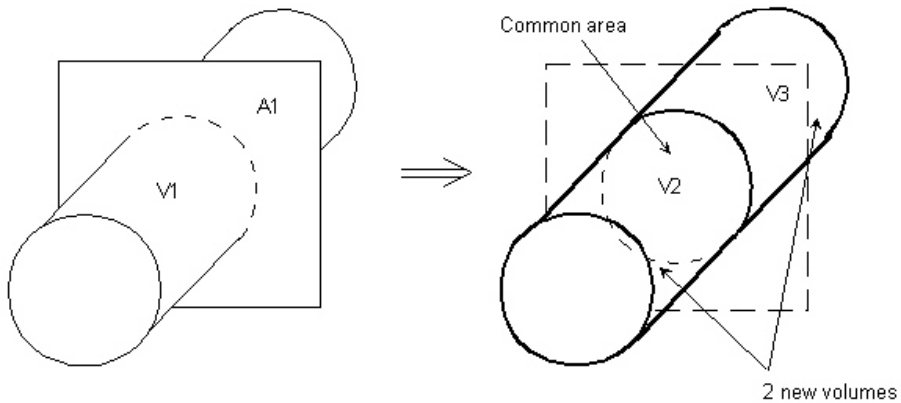


Booleans - Subtract

ASBL (Area Subtract Line)

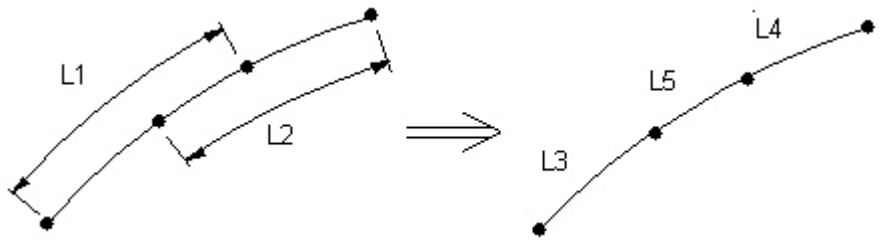


VSBA (Volume Subtract Area)

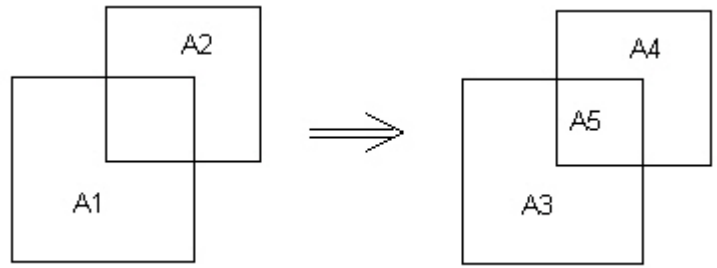


Booleans - Overlap

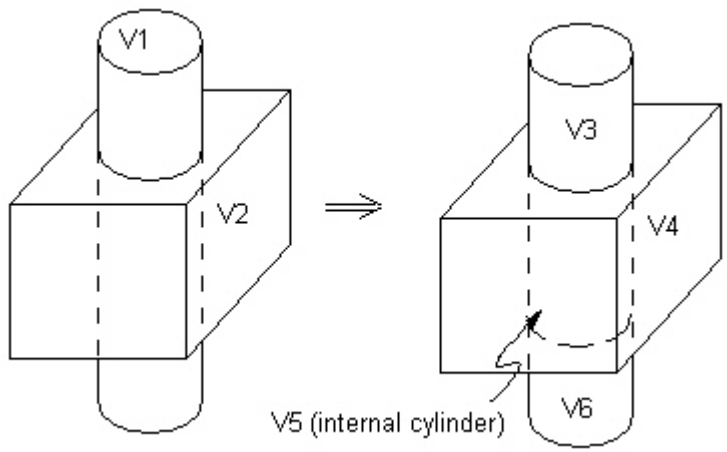
LOVLAP (Line Overlap Line)



AOVLAP (Area Overlap Area)

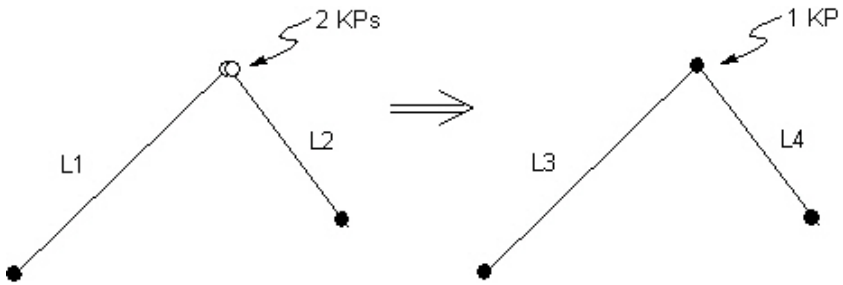


VOVLAP (Volume Overlap Volume)

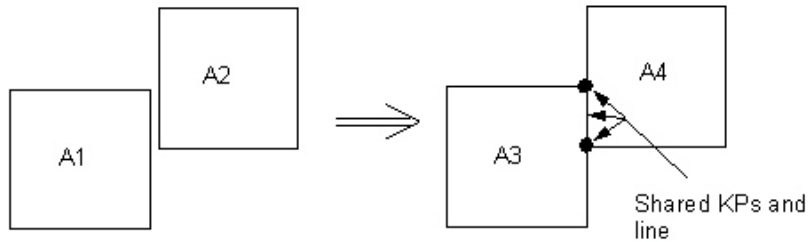


Booleans - Glue

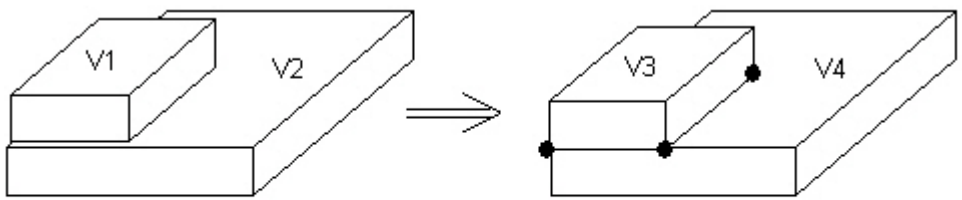
LGLUE (Line Glue Line)



AGLUE (Area Glue Area)



VGLUE (Volume Glue Volume)



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