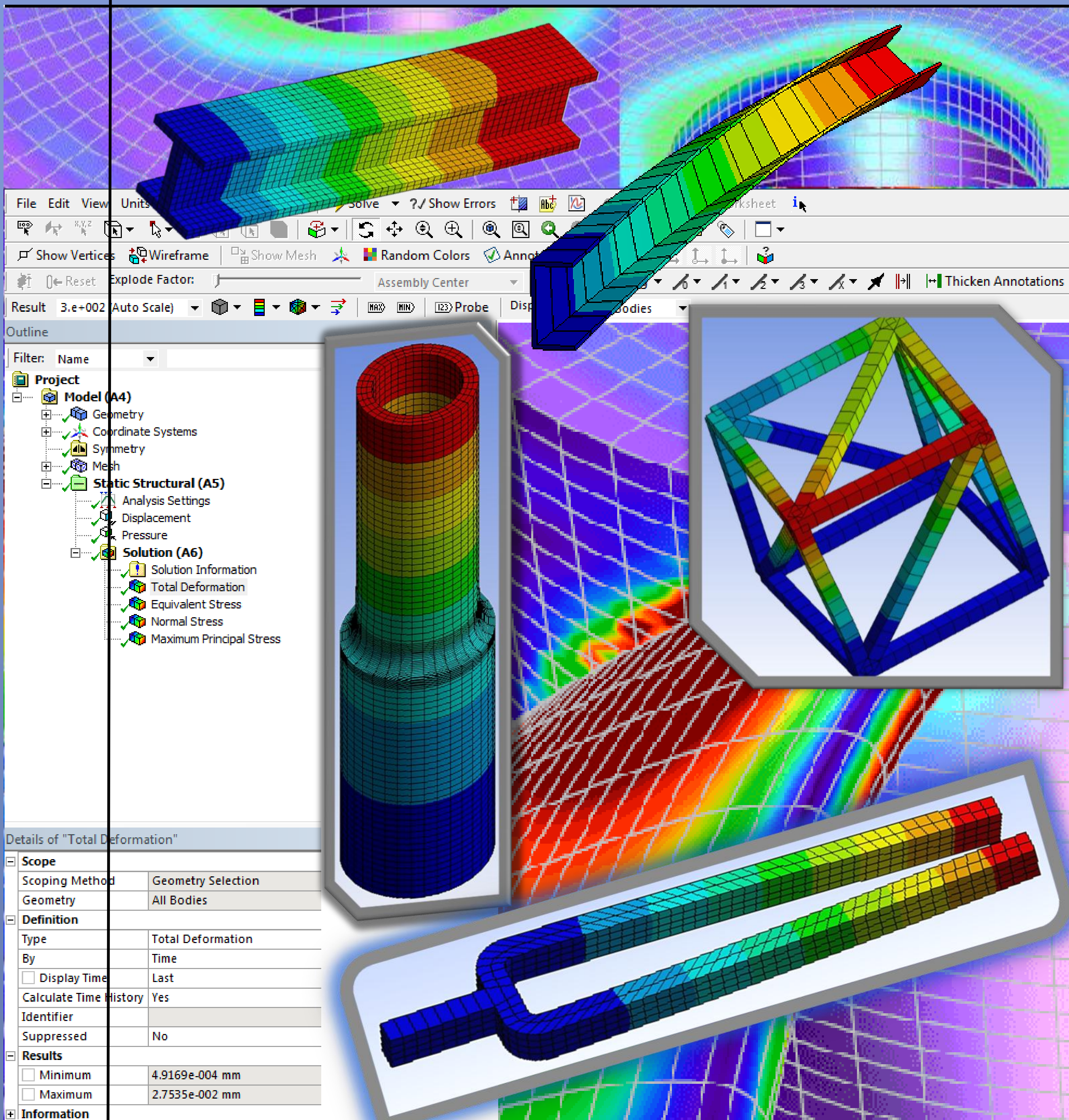
FINITE ELEMENT ANALYSIS

Guide through to ANSYS Workbench v16.2



ANSYS®
SMART ENGINEERING SIMULATION

Stefanos Syllignakis
Petr Vosynek

Finite Element Analysis Method using

ANSYS Workbench v16.2

Step-by-Step Guide…

Finite Element Analysis Method using ANSYS Workbench v16.2

Step~by~Step Guide…

Credits and Copyright

Written by: Bc. Syllignakis Stefanos
sylst3f@gmail.com

Main Editor: Ing. Petr Vosynek, Ph.D
petr.vosynek@gmail.com

Review and Editor: Ing. Marek Benešovský
*******@vutbr.cz

Preface

The presented material was created within the Erasmus+ project of the student Stefanos Syllignakis under the leadership of Petr Vosynek. It is basically support material for the subject 6KP and its English version 6KP-A (basics of computational modeling using finite element method) taught in the Institute of Solid Mechanics, Mechatronics and Biomechanics, Faculty of Mechanical Engineering, Brno University of Technology.

Computer labs of 6KP and 6KP-A are composed of active exercises under the current interpretation of the fundamentals associated with the type of elements and also from a separate project for a group of students. The texts were made in two versions, for the computing open_source system Salome_Meca (C_A) and for computing system ANSYS Workbench v16.2.

Table of Contents

INTRODUCTORY

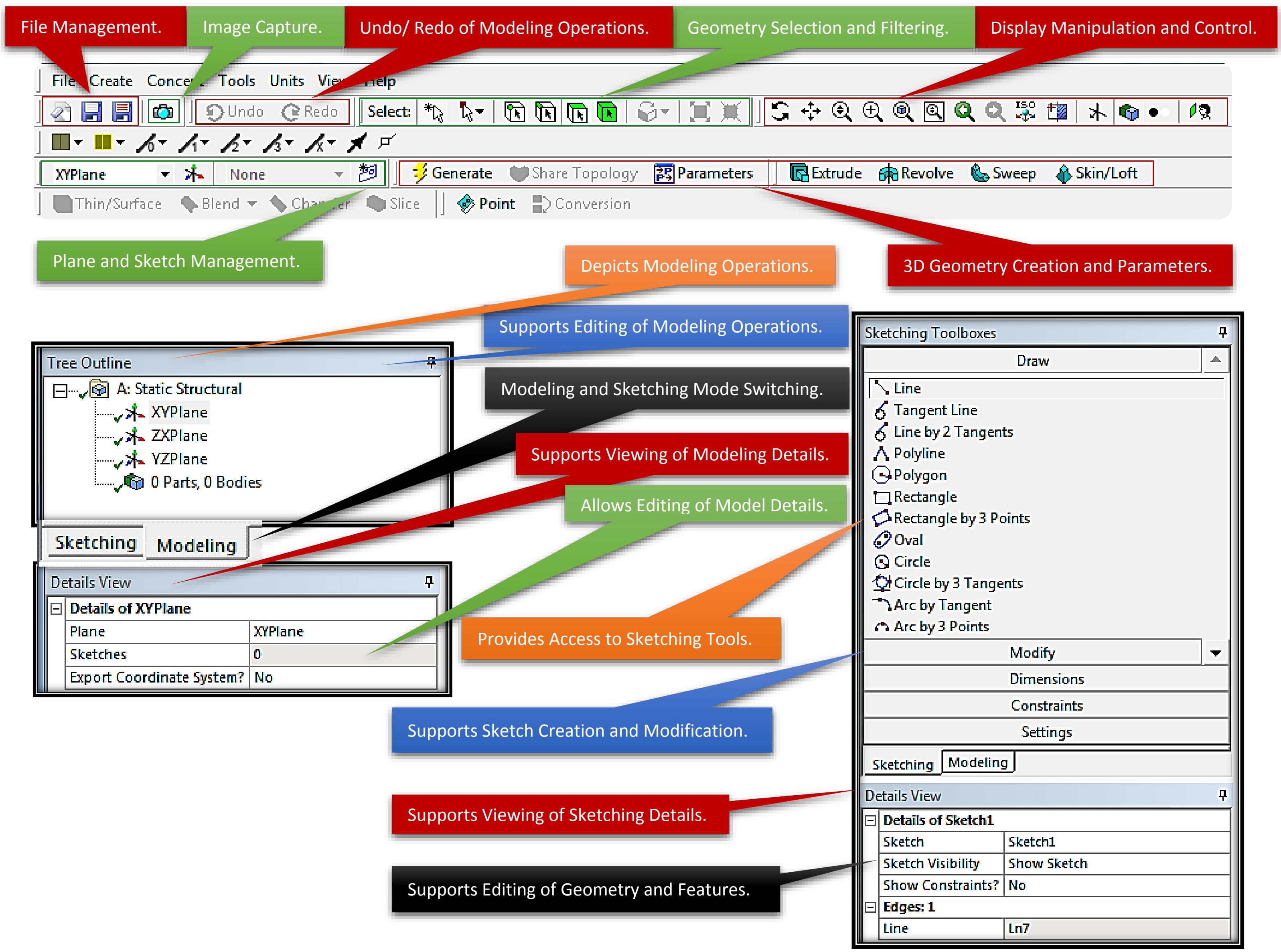
General Information

Reference --> *Autodesk Network Article.*

The ANSYS Workbench represents more than a general purpose engineering tool.
- It provides a highly integrated engineering simulation platform.
- It supports multi physics engineering solutions,
- It provides bi-directional parametric associativity with most available CAD systems.

These tutorials are designed to introduce you to
- The capabilities, functionalities and features of the ANSYS Workbench.
- The nature and design of the ANSYS Workbench User Interface.
- The concepts of ANSYS Workbench Projects and related engineering simulation capabilities.
- The integrated nature of ANSYS Workbench technology.
- The power of the ANSYS Workbench in using applied parametric modeling and simulation techniques to provide quality engineering solutions.

The Design Modeler

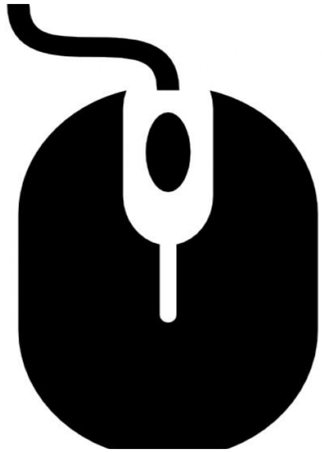


Sketching Mode:
- Provides for the creation of sketches using standard or user defined model coordinate systems.
- Supports the creation of 3D parametric solids from 2D sketches.

Modeling Mode:
- Provides tools for the creation and modification of 3D parts and models.
- Tracks and supports modification of modeling operations.

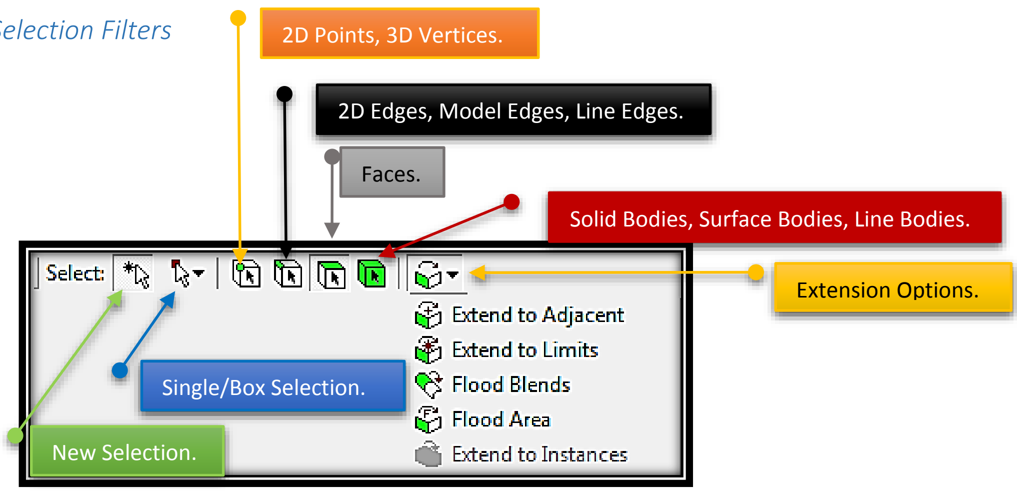
Basic Mouse Functionality

- Left Mouse Button. [LMB]
  - Geometry Selection.
  - Ctrl+ LMB → Adds/ Removes Selected Entities.
  - Hold LMB and Sweep Cursor → Continuous Selection.
  - Press+ Hold → "Paint Select".

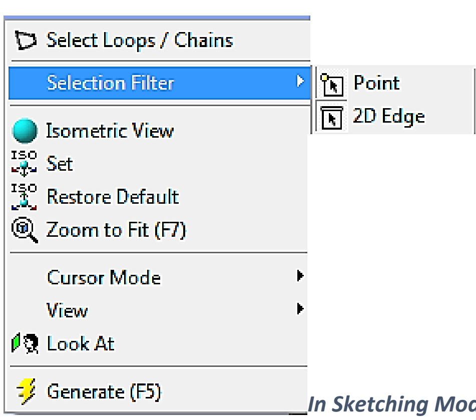


- Right Mouse Button. [RMB]
  - Open Pop-Up Context Menus.
- Middle Mouse Button. [MMB]
  - Free Rotation.

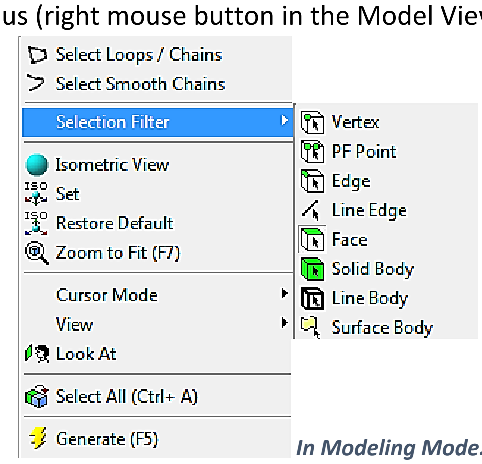
Selection Filters



- Model features are identified by graphically picking them using the left mouse button.
- Feature selection is done by activating one of the selection filters from the menu bar or from pop-up menus using the right mouse button.
- In selection mode, the cursor changes to reflect current selection filter.
- Adjacent and Flood Selections extend selections to adjacent areas. Additional information can be found in the ANSYS Workbench Help (documentation).
- Selection filters can also be set using pop-up menus (right mouse button in the Model View).

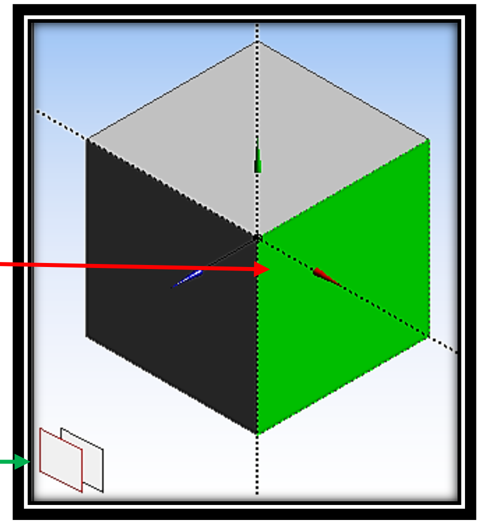


In Sketching Mode.



In Modeling Mode.

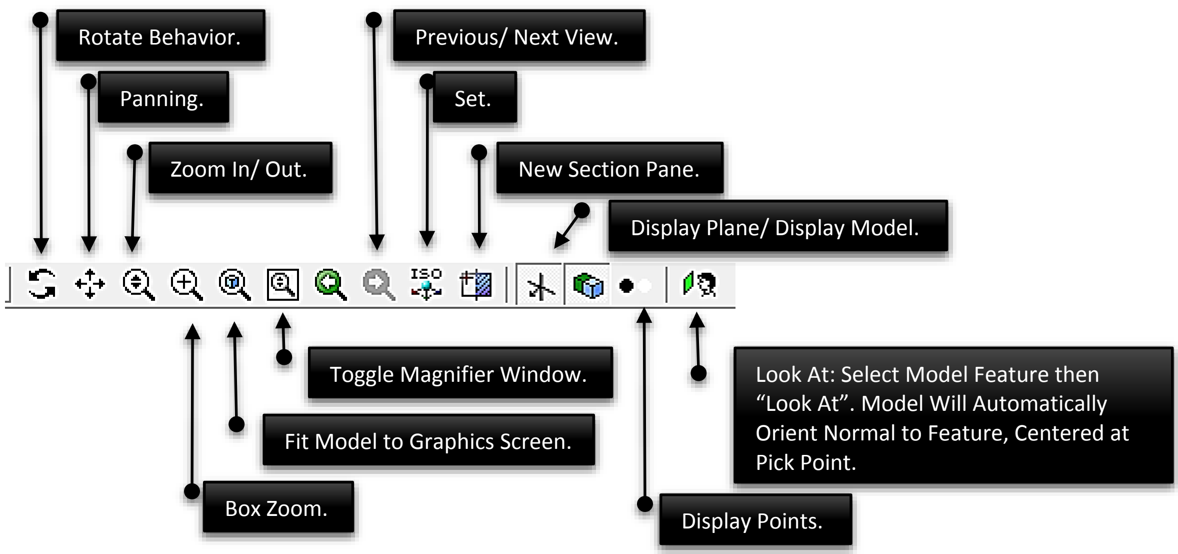
Selection Panes



- Selection Panes allow selecting hidden geometry (lines, surfaces, etc.) after an initial selection.
  - In assemblies only panes are color coded to match part colors.
  - Multi-select techniques apply to selection panes as well.
- Initial left mouse click.

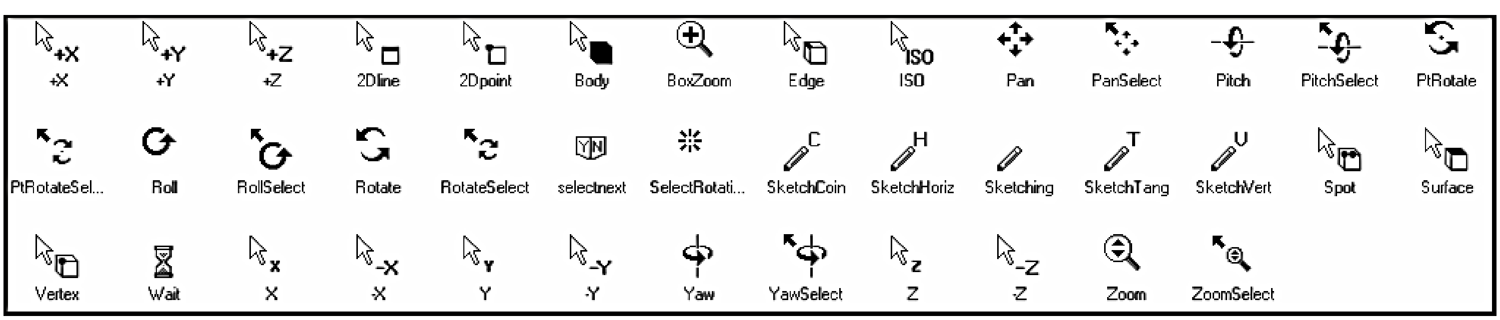
Note: *Each plane represents an entity (surface, edge, etc.) that an imaginary line would pass through, starting from the initial mouse click location and proceeding into the screen away from the viewer in the normal viewing direction.*

Graphic Controls



Additional Mouse Controls

- While in Select Mode:
  - Center Mouse Button → Free Rotations.
  - Right Mouse Button → Box Zoom.
  - Shift+ Center Mouse Button → Zoom.
- While in Rotate, Pan or Zoom Mode:
  - Left click on model temporarily resets center of view and rotation at cursor location.
  - Left click in open area re-centers model and rotation center to centroid.
- Mouse Cursor is Context Sensitive.
  - Indication Current Mouse Actions [Viewing, Rotation, Selecting, Sketch AutoConstrains, etc.]



Understanding Cell States

ANSYS Workbench integrates multiple applications into a single seamless project flow, where individual cells can obtain data from and provide data to other cells. As a result of this flow of data, a cell's state can change in response to changes made up to the project. ANSYS Workbench provides visual indications of a cell's state at any given time via icons on the right side of each cell.

Reference --> *ANSYS Help v.17.0.*

Cell states can be divided into the following categories:

i. Typical Cell States

- **Unfulfilled**

Required upstream data does not exist. Some applications may not allow you to open them with the cell in this state. For example, if you have not yet assigned a geometry to a system, all downstream cells will appear as unfulfilled, because they cannot progress until you assign a geometry.

- **Refresh Required**

Upstream data has changed since the last refresh or update. You may or may not need to regenerate output data. When a cell is in a Refresh Required state, you can Edit the cell, Refresh the data, Update Upstream Components, or Update the cell.

The advantage to simply refreshing a cell rather than performing a full update is that you can be alerted to potential effects on downstream cells before updating and can make any necessary adjustments. This option is especially useful if you have a complex system in which an update could take significant time and/or computer resources.

- **Attention Required**

All of the cell's inputs are current; however, you must take a corrective action to proceed. To complete the corrective action, you may need to interact with this cell or with an upstream cell that provides data to this cell. Cells in this state cannot be updated until the corrective action is taken.

This state can also signify that no upstream data is available, but you can still interact with the cell. For instance, some applications support an "empty" mode of operation, in which it is possible to enter the application and perform operations regardless of the consumption of upstream data.

- **Update Required**

Local data has changed and the output of the cell needs to be regenerated.

- **Up to Date**

An Update has been performed on the cell and no failures have occurred. It is possible to edit the cell and for the cell to provide up-to-date generated data to other cells.

- **Input Changes Pending**

The cell is locally up-to-date but may change when next updated as a result of changes made to upstream cells.

ii. Solution-Specific States

- **Interrupted, Update Required**

Indicates that you have interrupted the solution during an update, leaving the cell paused in an Update Required state.

This option performs a graceful stop of the solver, which will complete its current iteration; although some calculations may have been performed, output parameters will not be updated. A solution file will be written containing any results that have been calculated. The solve will be resumed with the next update command.

- **Interrupted, Up to Date**

Indicates that you have interrupted the solution during an update, leaving the cell in an Up-to-Date state.

This option performs a graceful stop of the solver, which will complete its current iteration; output parameters will be updated according to the calculations performed thus far and a solution file will be written. You can use the solution for post processing (to look at the intermediate result, for example). Because the cell is already up-to-date, it will not be affected by a design point update; to resume the solve, right-click and select the **Continue Calculation** option.

- **Pending**

Signifies that a batch or asynchronous solution is in progress. When a cell enters the Pending state, you can interact with the project to exit ANSYS Workbench or work with other parts of the project. If you make changes to the project that are upstream of the updating cell, then the cell will not be in an up-to-date state when the solution completes.

iii. Failure States

- **Refresh Failed, Refresh Required**

The last attempt to refresh cell input data failed and the cell remains in a refresh required state.

- **Update Failed, Update Required**

The last attempt to update the cell and calculate output data failed and the cell remains in an update required state.

- **Update Failed, Attention Required**

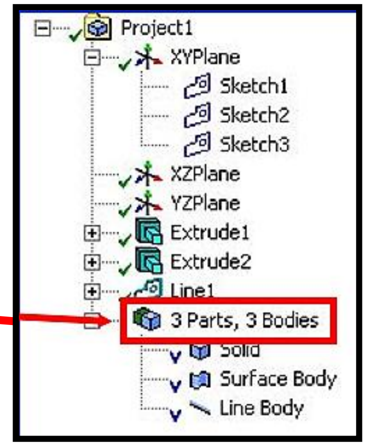
The last attempt to update the cell and calculate output data failed. The cell remains in an attention required state.

If an action results in a failure state, you can view any related error messages in the **Messages** view by clicking the **Show Messages** button on the lower right portion of the ANSYS Workbench tab.

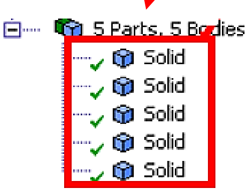
3D Geometry

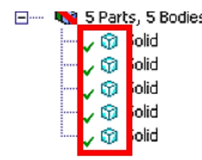
Reference --> *Autodesk Network Article.*

Bodies and Parts



- # DesignModeler is primarily intended to provide geometry to an analysis environment. For this reason we need to see how DesignModeler treats various geometries.
- # DesignModeler contains three different body types:
  - Solid Body: A body with surface area and volume.
  - Surface Body: A body with surface area but no volume.
  - Line Body: A body which consists entirely of edges, no area, and no volume.
- # By default, DesignModeler places each body into one part by itself.
- # There are two body types in DesignModeler:
  - Active:
    - Body can be modified by normal modeling operations.
    - Active bodies are displayed in blue in the Feature Tree View.
    - The body's icon in the Feature Tree View is dependent on its type – solid, surface, or line.
  - Frozen:
    - Two Purposes:
      - Provides alternate method for Sim Assembly Modeling.
      - Provides ability to "Slice" parts.
    - A Frozen body is immune to all modeling operations except slicing.
    - To move all Active bodies to the Frozen state, use the Freeze feature.
    - To move individual bodies from the Frozen to Active, select the body and use the Unfreeze feature.
  - Frozen bodies are displayed "lighter" in the Tree View.

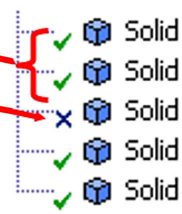




- # Body Suppression:
  - Suppressed bodies are not plotted.
  - Suppressed bodies are not sent to Design Simulation for analysis, nor are they included in the model when exporting to a Parasolid (.x_t) or ANSYS Neutral File (.anf) format.
  - In the Tree View an "X" is shown near suppressed bodies.

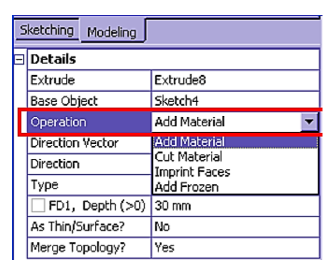
Unsuppressed

Suppressed



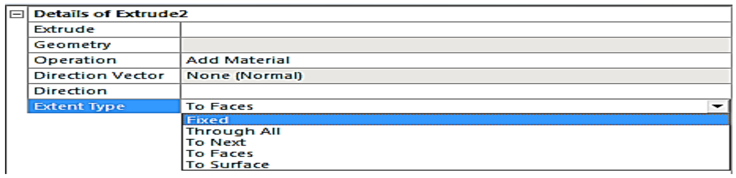
- # Parts:
  - By default, the DesignModeler places each body into one part by itself.
  - You can group bodies into parts.
    - These parts will be transferred to Design Simulation as parts consisting of multiple bodies (volumes), but **Shared Topology**.
  - To form a new part, select two or more bodies from the graphics screen and use → Tools → Form New Part.
  - The Form New Part option is available only when bodies are selected and you are not in a feature creation or feature edit state.

Boolean Operations

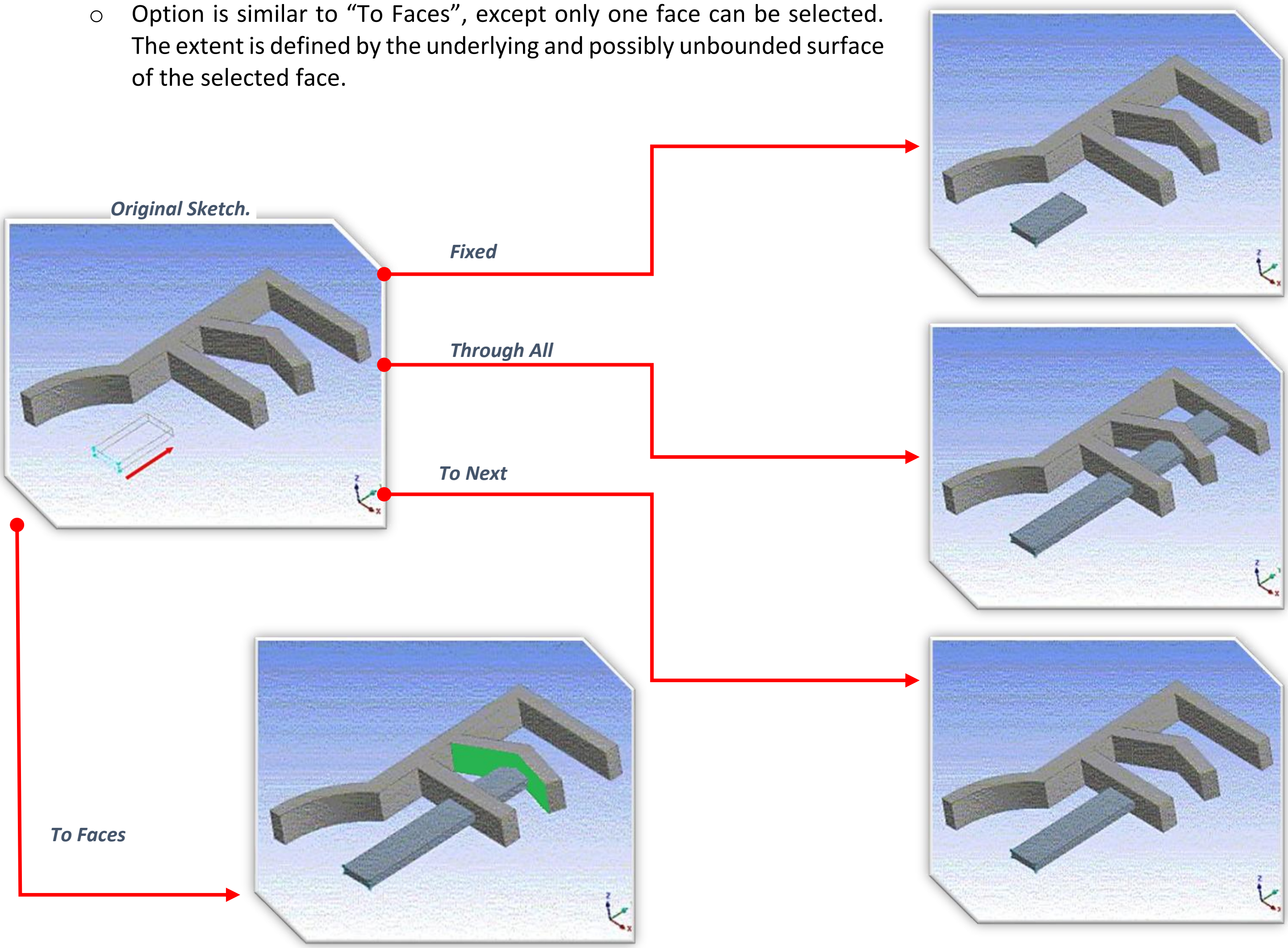


- # You can apply five different Boolean operations to 3D features:
  - Add Material: Creates material and merges it with the active bodies.
  - Cut Material: Removes material from active bodies.
  - Slice Material: Slices frozen bodies into pieces. [Available only when all bodies in the model are frozen]
  - Imprint Faces: Similar to Slice, except that only the faces of the bodies are split, and edges are imprinted if necessary.
  - Add Frozen: Similar to Add Material, except that the feature bodies are not merged with the existing model but rather added as frozen bodies.

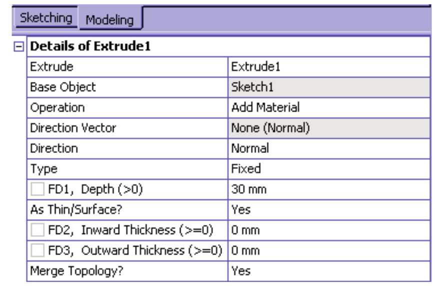
Feature Type



- # Fixed:
  - Fixed extents will extrude the profiles the exact distance specified by the depth property. The feature preview shows an exact representation of how the feature will be created.
- # Through All Type:
  - Will extend the profile through the entire model.
  - When adding material the extended profile must fully intersect the model.
- # To Next:
  - Add will extend the profile up to the first surface it encounters.
  - Cut, Imprint and Slice will extend the profile up to and through the first surface or volume it encounters.
- # To Faces:
  - Allows you to extend the Extrude feature up to a boundary formed by one or more faces.
  - For multiple profiles make sure that each profile has at least one face intersecting its extent. Otherwise, an extent error will result.
  - The “To Faces” option is different from “To Next”. To Next does not mean “to the next face”, but rather “through the next chunk of the body”.
  - The “To Faces” option can be used with respect to faces of frozen bodies.
- # To Surface:
  - Option is similar to “To Faces”, except only one face can be selected. The extent is defined by the underlying and possibly unbounded surface of the selected face.

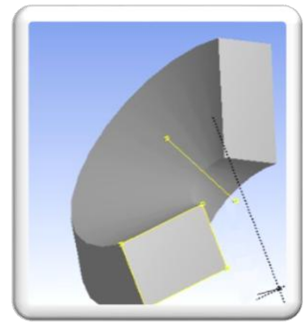


Feature Creation



Details of Extrude1	
Extrude	Extrude1
Base Object	Sketch1
Operation	Add Material
Direction Vector	None (Normal)
Direction	Normal
Type	Fixed
FD1, Depth (>0)	30 mm
As Thin/Surface?	Yes
FD2, Inward Thickness (>=0)	0 mm
FD3, Outward Thickness (>=0)	0 mm
Merge Topology?	Yes

- # Extrusions:
  - Extrusions include solids, surfaces and thin walled features.
  - To create surfaces, select "as thin/surface" and set the inner and outer thickness to zero.
  - The active sketch is the default input but can be changed by selecting the desired sketch in the Tree View.
  - The Detail View is used to set the Extrude depth, direction and Boolean Operation (Add, Cut, Slice, Imprint or Add Frozen).
  - The Generate button completes the feature creation.
- # Revolve:
  - Active sketch is rotated to create 3D Geometry.
  - Select axis of rotation from details.
  - Direction Property for Revolve:
    - Normal: Revolves in positive Z direction of base object.
    - Reversed: Revolves in negative Z direction of base object.
    - Both- Symmetric: Applies feature similar in both directions.
    - Both- Asymmetric: Applies feature in both directions unevenly.
  - The Generate button completes the feature creation.



- # Sweep:
  - Solids, Surfaces and thin walled features can be created by using this feature to sweep a profile along a path.
  - Scale and Turns properties can be used to create helical sweeps.
    - Scale: Tapers or expands the profile along the path of the sweep.
    - Turns: Twists the profile as sweeps along the path.
    - A negative value for Turns will make the profile rotate about the path in the opposite direction.
  - Alignment:
    - Path Tangent: Reorients the profile as it is swept along the path to keep the profile in the path's tangent direction.
    - Global: The profile's orientation remains constant as it is swept along the path, regardless of the path's shape.
- # Skin/ Loft:
  - Takes a series of profiles from different planes to create 3D Geometry fitting through them.
    - A profile is a sketch with one closed or open loop or a plane from a face.
    - All profiles must have the same number of edges.
    - Open and closed profiles cannot be mixed.
    - All profiles must be of the same type.
  - Sketches and planes can be selected by clicking on their edges or points in the graphics area, or by clicking on the sketch or plane in the feature tree.
  - After selecting an adequate number of profiles, a preview will appear showing he selected profiles and the guide polygon.
  - The guide polygon is a gray poly-line which shows how the vertices between the profiles will line up with each other.
  - Skin/ Loft operation relies heavily on Right Mouse Button menu choices.