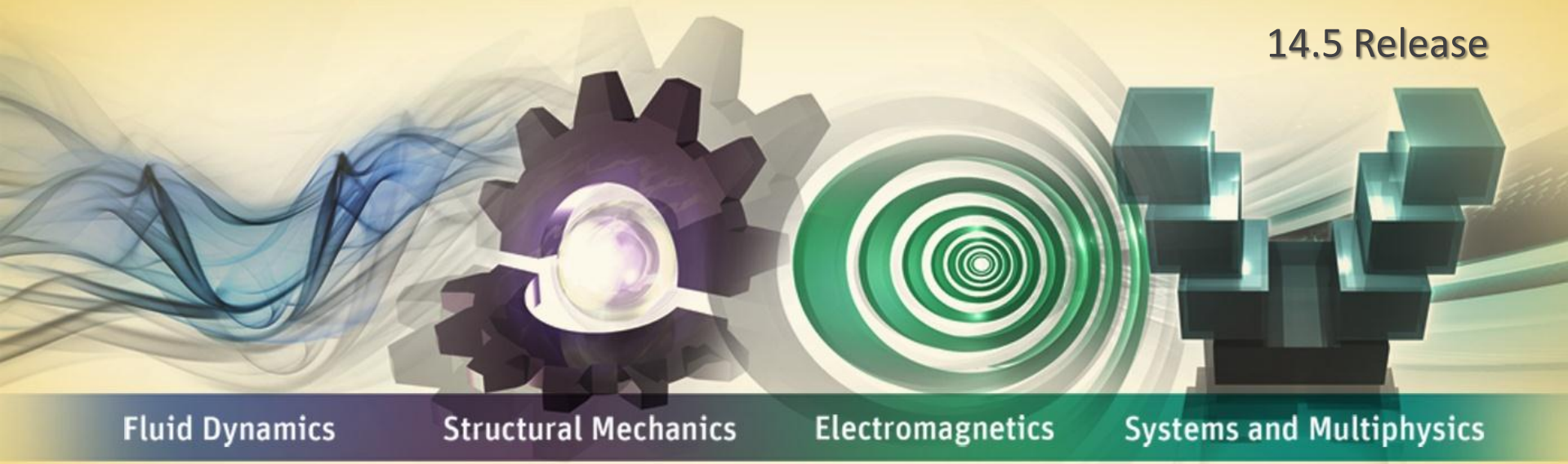
ANSYS

Realize Your Product Promise™

Lecture 1
Introduction to ANSYS ICEM CFD

14.5 Release



Introduction to ANSYS ICEM CFD

Purpose/Goals

Ansys ICEM CFD is a general purpose grid generating program

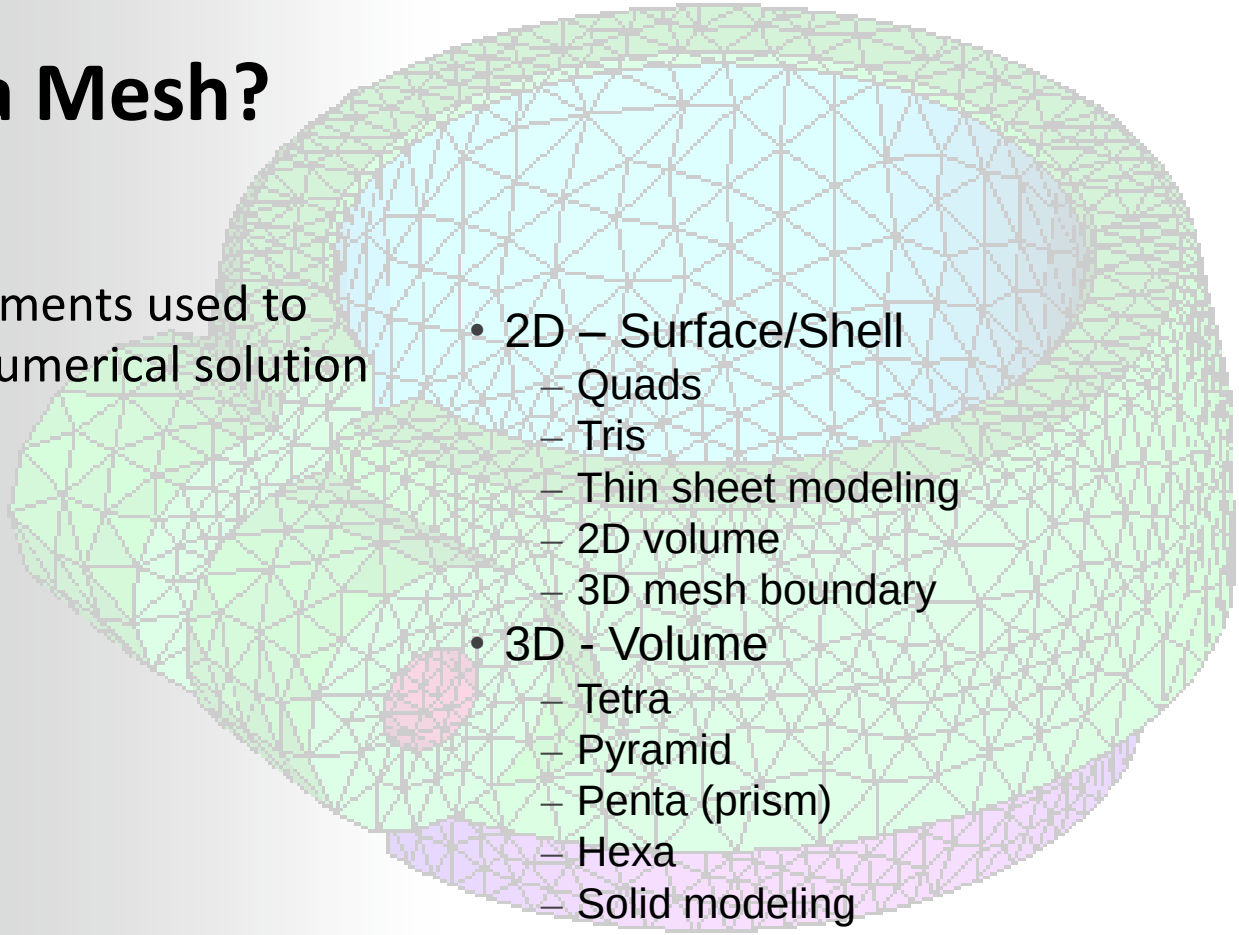
- Grids for various solver types can be produced
  - Fluid analysis (CFD)
  - Structural analysis (FEA)
  - Electromagnetic, and more
- The terms “mesh,” “grid,” and “discretizing the domain” all refer to the same thing

The purpose of this course is to teach the basic tools and methods for generating meshes with Ansys ICEM CFD

- Geometry import/editing
- Meshing (surface meshing, tetra, prism, hexa, hybrid meshes)
- Mesh Editing (smoothing, error checks, individual element editing)
- Solver setup

Best meshing practices will be shown as well as the tools needed to produce them

What is a Mesh?

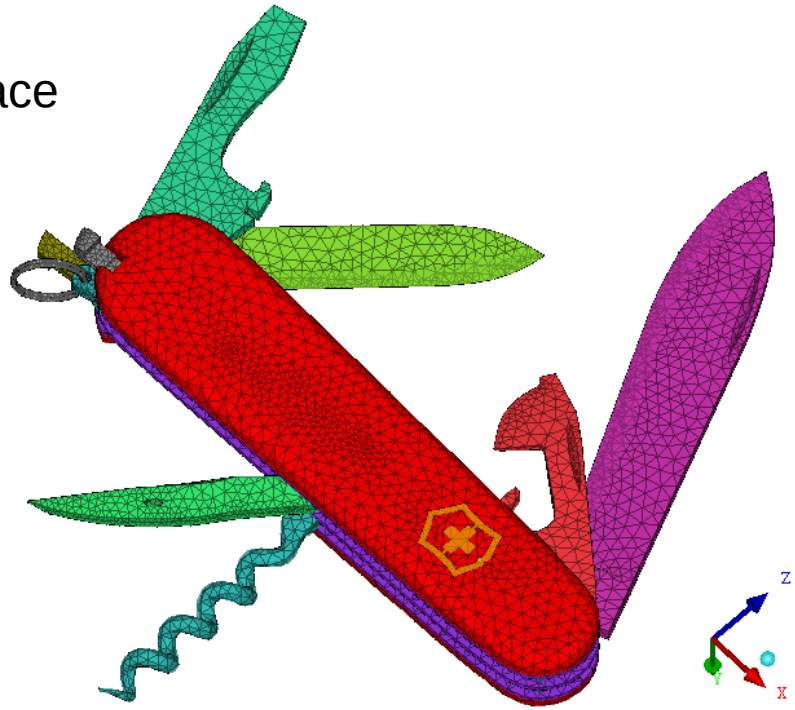


Mesh

- Volume comprised of elements used to discretize a domain for numerical solution
  - Structural
  - Fluid dynamics
  - Electromagnetics
  - Other
- Elements
  - 0D – Node element
    - Point mass
    - Constraint, load location
  - 1D – Lines
    - Bars, beams, rods, springs
    - 2D mesh boundary
  - 2D – Surface/Shell
    - Quads
    - Tris
    - Thin sheet modeling
    - 2D volume
    - 3D mesh boundary
  - 3D - Volume
    - Tetra
    - Pyramid
    - Penta (prism)
    - Hexa
    - Solid modeling
    - 3D fluid modeling
  - Formats
    - Unstructured
    - Block Structured
  - Nodes
    - Point locations of element corners

Ansys ICEM CFD Features

- **Geometry Creation/Repair/Simplification**
  - Including Mid-Plane Extractions/Extensions
  - Most geometry intended to be imported
- **Powerful Meshing tools**
  - Tetra/Prism from CAD and/or existing surface mesh
  - Shell meshing: structured, unstructured
  - Hex-dominant, swept, Structured hexa, Extruded quads, Body-fitted and stair-step Cartesian, hexa-core
  - Hybrid meshing (merging, multi-zone)
- **Advanced mesh editing**
- **Solver Setup**
- **Output to 100+ Solvers**
- **Scripting … and much more…**



Ansys ICEM CFD Product Purchasing

ICEM CFD

- Full capabilities (all meshers, CFD and FEA)
- Enables Ansys meshing in Workbench

ICEM CFD Tetra/Prism

- Includes hexa core, but not other hexa methods
- Enables Ansys meshing in Workbench

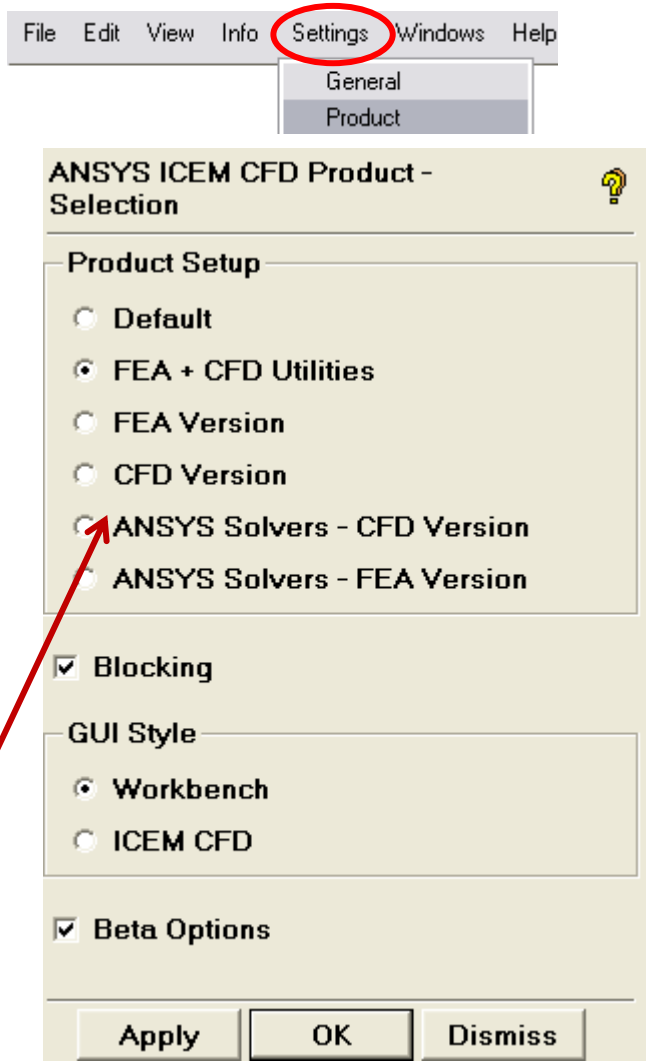
ICEM CFD hexa

- No tetra/prism
- Includes Body-fitted Cartesian (BFC) and Hex-dominant

ICEM CFD Add-on

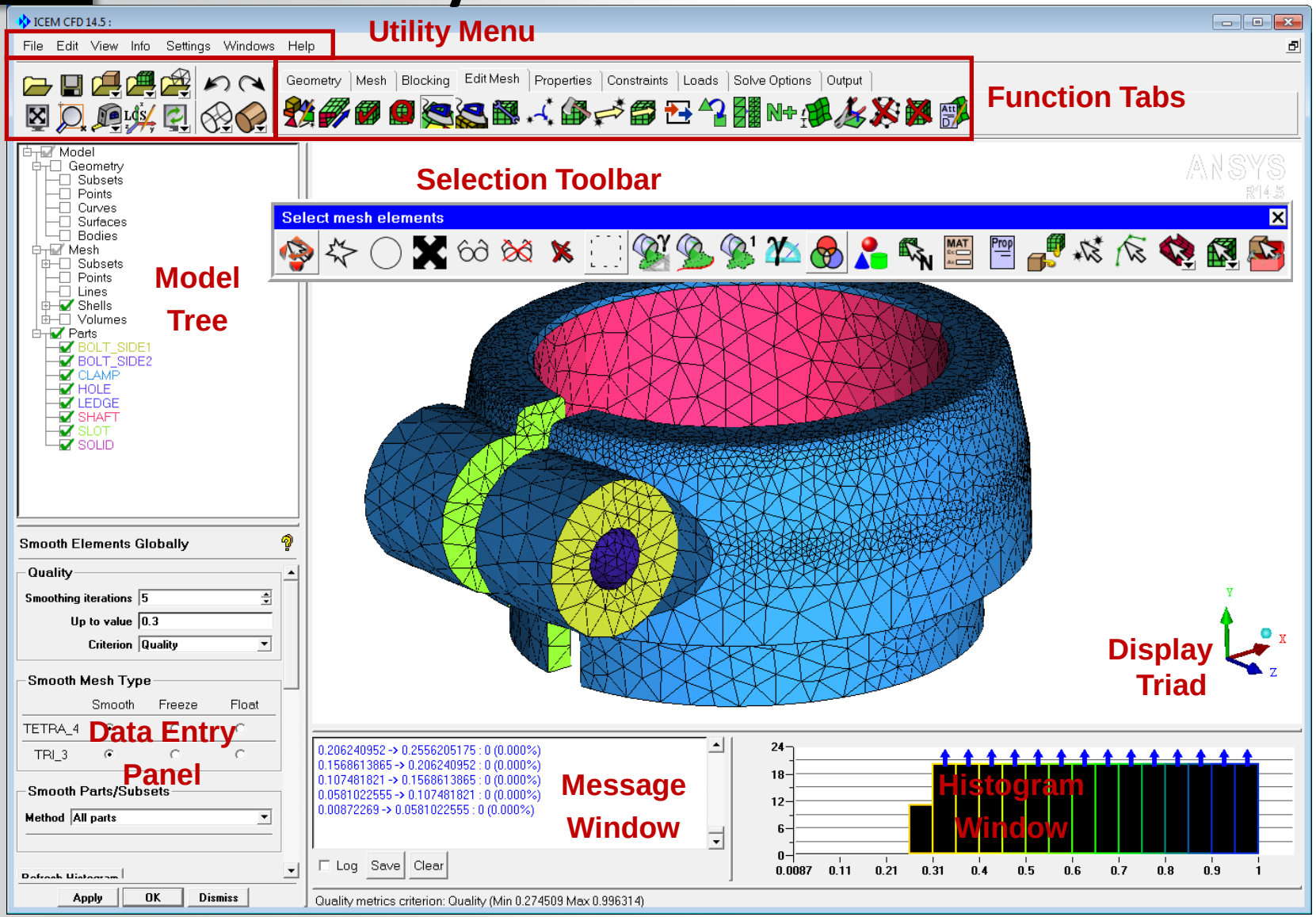
- Lower cost, added to Ansys solver bundles
- Full capabilities
- Only writes to Ansys solvers (Ansys, CFX, Fluent)

CAD interfaces sold separately



Settings->Product filters what options are seen in the application, but the licensing determines which ones can be used

GUI and Layout



File and Directory Structure

Use of many files

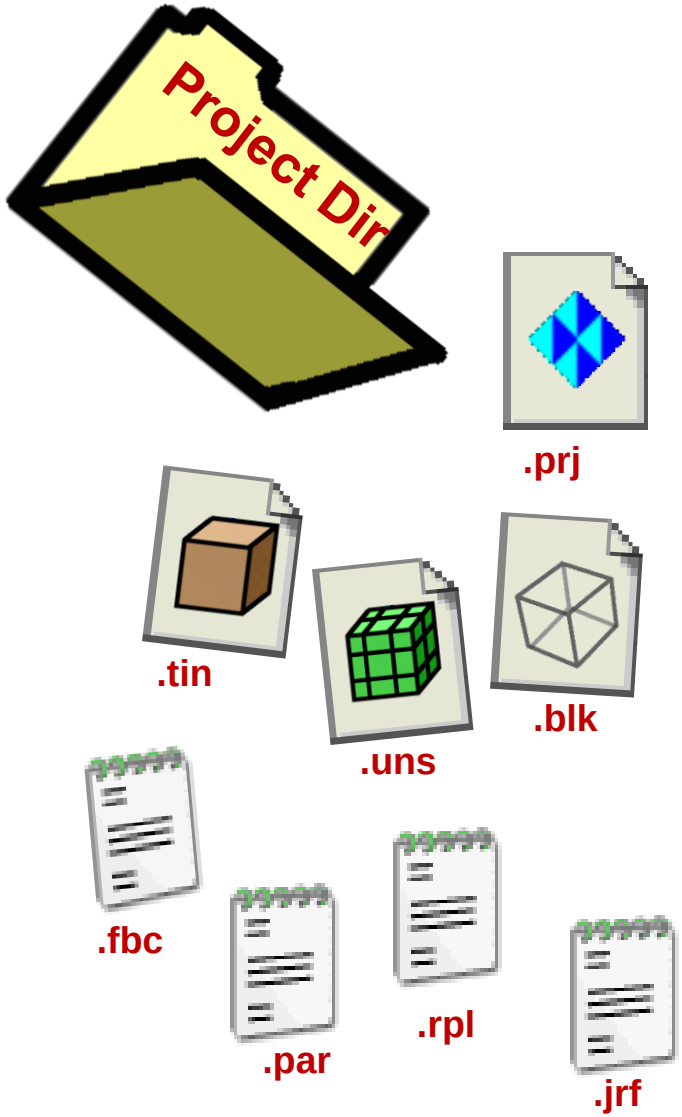
- Not one large common database
- For faster input/output

All files can optionally be associated within a Project

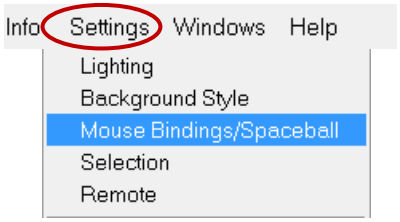
- Establishes working directory when opened or saved
- Settings (***.prj**) file contains associated file names

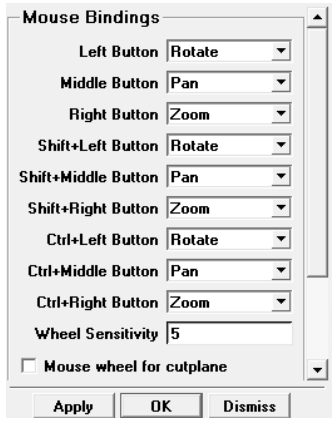
Primary file types:

- **Tetin (.tin):** Geometry
  - Geometry entities and material points
  - Part associations
  - Global and entity mesh sizes
  - Created in Ansys ICEM CFD or CAD Interface
- **Domain file (.uns)**
  - Unstructured mesh
- **Blocking file (.blk)**
  - Blocking topology
- **Attribute file (.fbc, .atr)**
  - Boundary conditions, local parameters & element types
- **Parameter file (.par)**
  - solver parameters & element types
- **Journal and replay file(.jrf, .rpl)**
  - Record of performed operations (echo file)



Mouse Usage



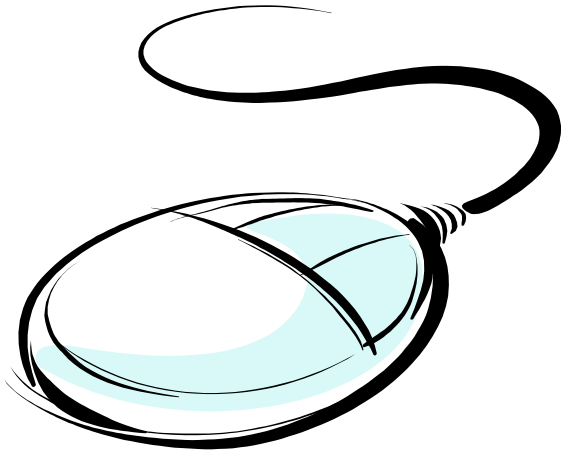


'Dynamic' viewing mode (click and drag)

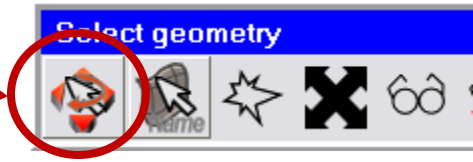
- left: rotate (about a point)
- middle: translate
- right: zoom (up-down)
  screen Z-axis rotation (sideways)
- Wheel zoom

Selection mode (click)

- left: select (click and drag for box select)
- middle: apply operation
- right: unselect last selection



- ***F9*** **toggles the mouse control to Dynamic mode while in Select mode**
  - ***Toggle Dynamics*** button also does this



- **Spaceball allows for dynamic motion even while in selection mode**

Utility Menus

File Menu (file i/o)

- New Project...
- Open Project...
- Save Project...
- Save Project As...
- Close Project...
- Change Working Dir...
- Geometry ▸
- Mesh ▸
- Blocking ▸
- Attributes ▸
- Parameters ▸
- Cartesian ▸
- Import Geometry ▸
- Import Mesh ▸
- Export Geometry ▸
- Export Mesh ▸
- Workbench Readers
- Replay Scripts ▸
- Exit

Edit Menu

- Undo
- Redo
- Clear Undo
- Shell
- Facets -> Mesh
- Mesh -> Facets
- Struct mesh->CAD Surfaces
- Struct mesh->Unstruct Mesh
- Struct mesh->Super Domain
- Shrink tetin file

View Menu

- Fit
- Box Zoom
- Top
- Bottom
- Left
- Right
- Front
- Back
- Isometric
- View Control ▸
- Save Picture
- Mirrors and Replicates
- Annotation ... ▸
- Add Marker
- Clear Markers
- Mesh Cut Plane

Info Menu

- Geometry Info
- Surface Area
- Frontal Area
- Curve Length
- Curve Direction
- Mesh Info
- Mesh Area/Volume
- Element Info
- Node Info
- Element Type / Property Info
- Toolbox
- Project File
- Domain File
- Mesh Report

Settings Menu (preferences)

- General
- Product
- Display
- Speed
- Memory
- Lighting
- Background Style
- Mouse Bindings/Spaceball
- Selection
- Remote
- Model
- Geometry Options
- Meshing Options ▸
- Solver
- Restore
- Reset

Help Menu

- Help Topics
- Tutorial Manual
- User Manual
- Programmer's Guide
- Output Interfaces
- Installation & Licensing Guide
- What's New
- Legal Notices
- Show Customer Number
- About ANSYS ICEM CFD

File menu

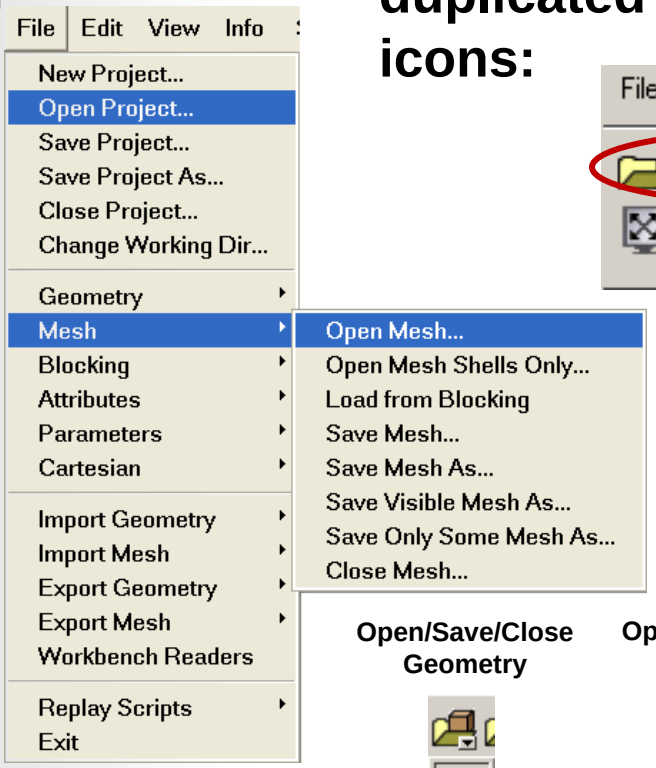
To open/save/close

- Projects
  - Will open/save/close all associated files including
    - Geometry (*.tin)
    - Mesh (*.uns)
    - Blocking (*.blk)
    - Attributes… (*.fbc, *.atr)
    - Parameters (*.par)
- All file types can be opened/saved/closed independently

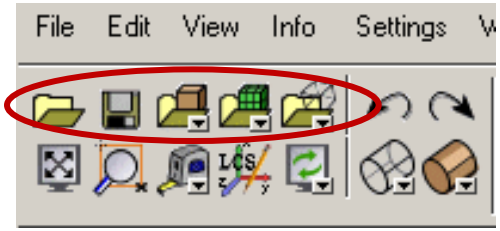
Also to

- Import/Export Geometry/Mesh
- Invoke scripting

Exit



- **Several common functions are duplicated as utility icons:**



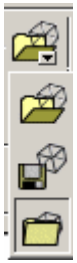
Open Project

Save Project

Open/Save/Close Geometry

Open/Save/Close Mesh

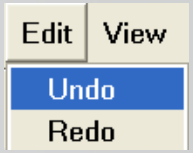
Open/Save/Close Blocking



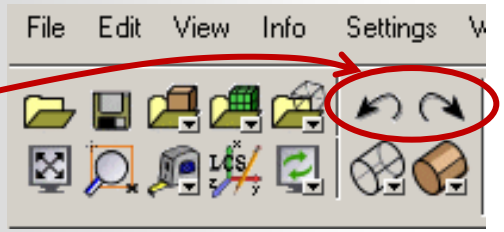
***Save* frequently to new file names so you can go back!**

Other Commonly Used Utilities

Edit > Undo/Redo



Also here



View

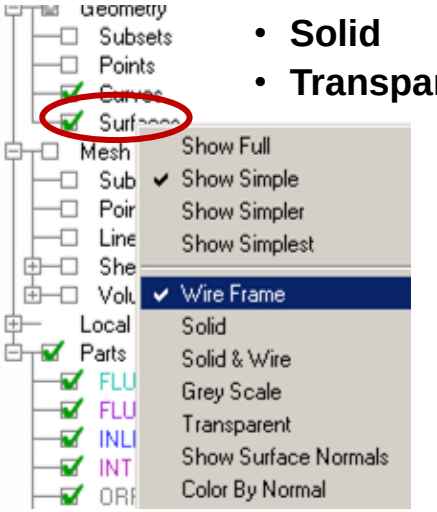
- Fit 
  - Fit visible entities into screen
- Box Zoom 
- Standard views
  - Top, Bottom, Left, etc.
  - Can also select X, Y, Z axis of display triad in lower right hand corner of main view screen to orient to standard views, e.g. selecting "X" will orient "right"
  - Isometric – select blue dot within triad

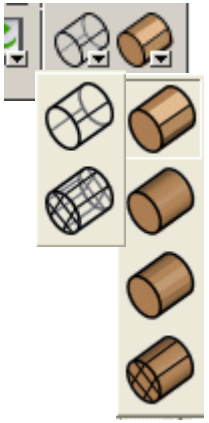
Measure

- Distance
- Angle
- Location

- Local Coordinate System 
  - Used by:
    - Any XYZ point location entry in menus
    - Select location
    - Measuring
    - Node/point movement/creation
    - Alignment
    - Loads
    - Transformation

- Surface display
  - Wireframe
  - Solid
  - Transparent





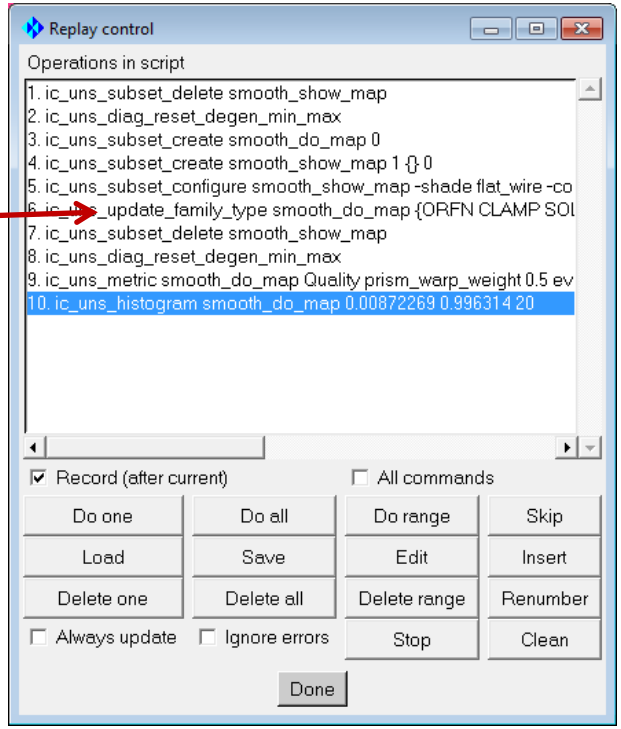
Scripting

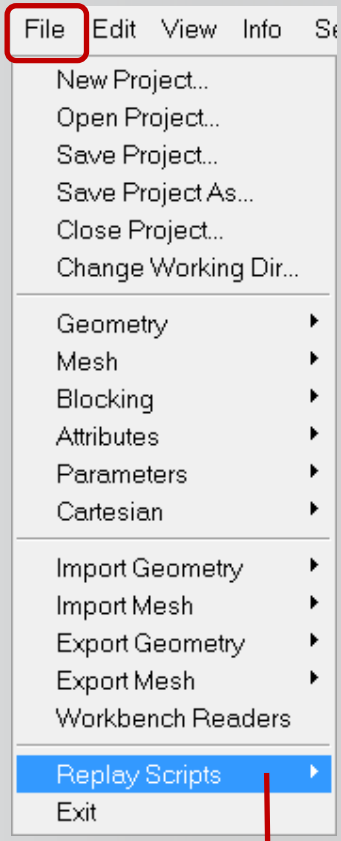
Scripts can be run and recorded from the File menu

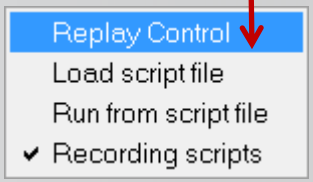
- ***Replay Control***
  - Opens a window to record and edit commands
  - Step through your process on a test model first to print the commands
  - Then delete the unnecessary commands
  - Edit the script such as replacing file names and command arguments with variables
- The command names are descriptive of the action

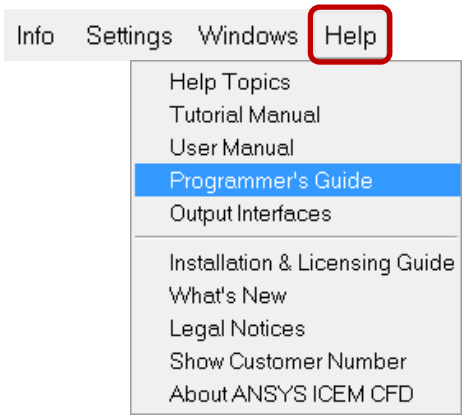
Help > Programmer's Guide **documents the commands**

- The first section gives you a quick start to scripting with ICEM CFD

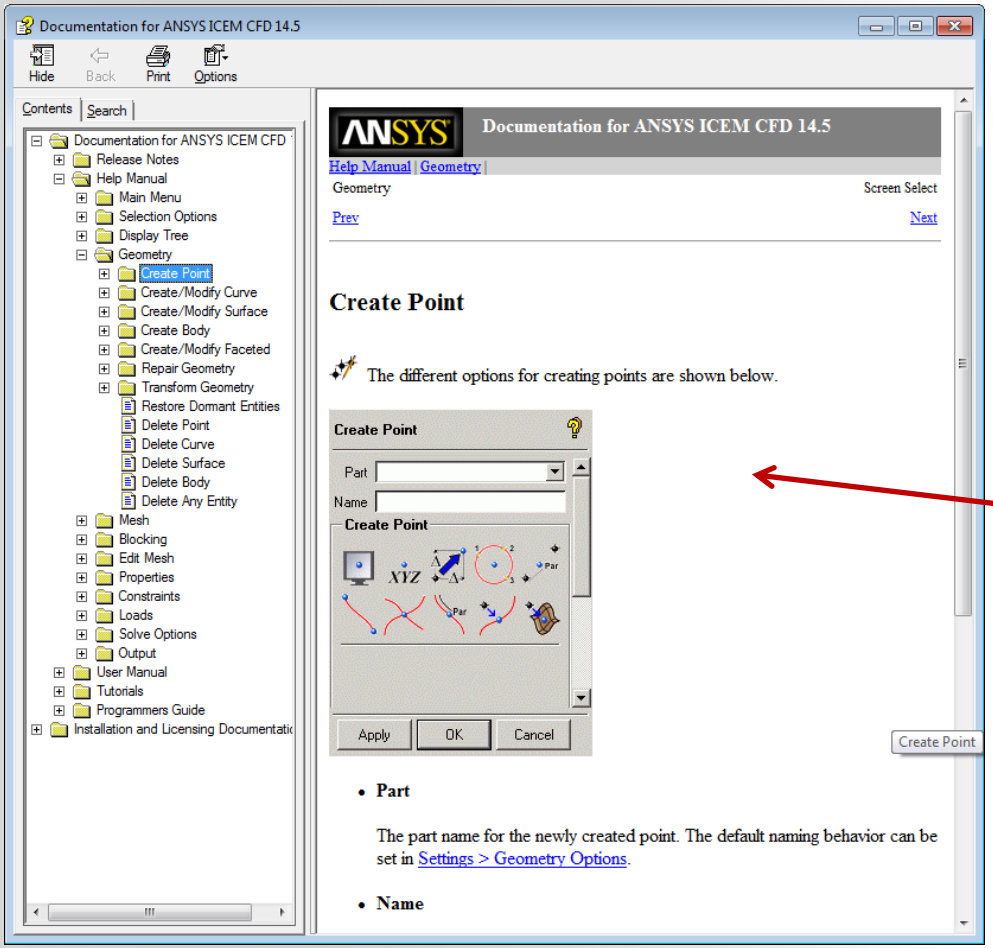


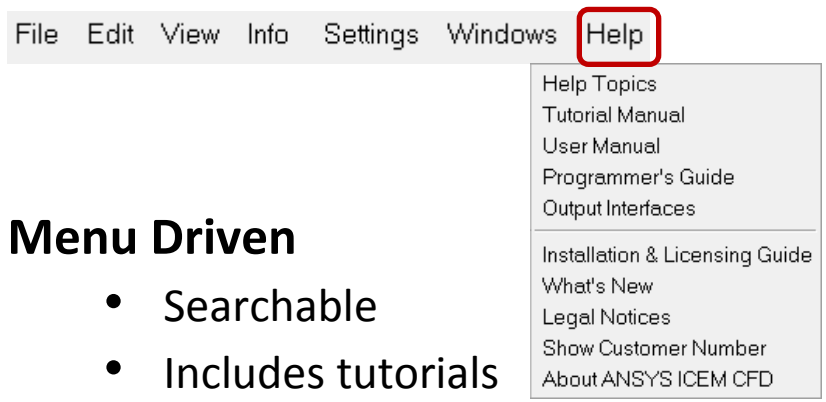






Help

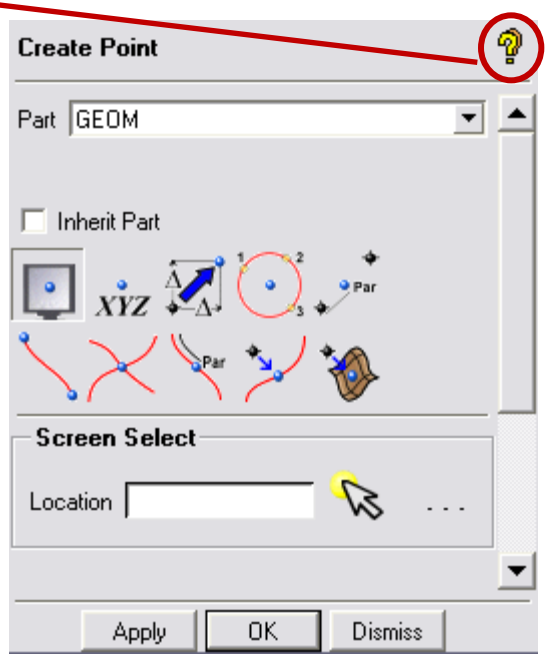




Menu Driven

- Searchable
- Includes tutorials
- Programmers guide (for ICEM CFD/Tcl scripting procedures)

Hyper-link to specific topic



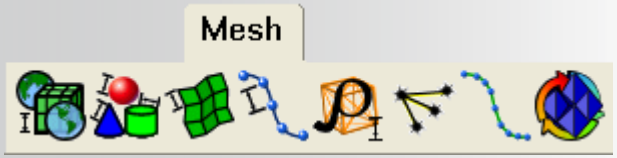
- **Bubble explanation with cursor positioning**

Common Function Tabs

Geometry

Create/Modify geometry

Mesh



Set mesh sizes, types and methods
Set global mesh options
Auto create Shell, Volume, Prism meshes

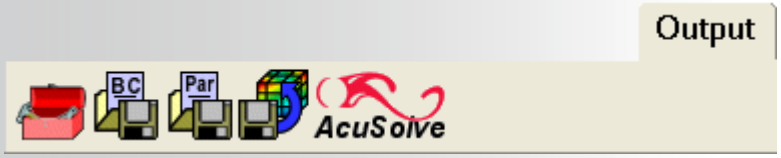
Blocking

Initialize blocking
Split/modify blocks
Generate structured hexa mesh

Edit Mesh

Check errors/problems, Smooth, Refine/Coarsen, Merge, repair mesh, Transform, etc.

Output



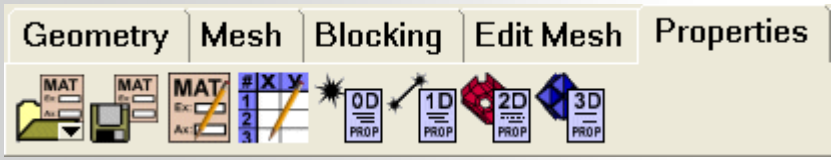
Set Boundary Conditions and Parameters
Write mesh for 100+ solvers.

Structural Function Tabs

Only available when *Common Structural Solver* is set to Abaqus, Ansys, Autodyn, LS-Dyna, or Nastran

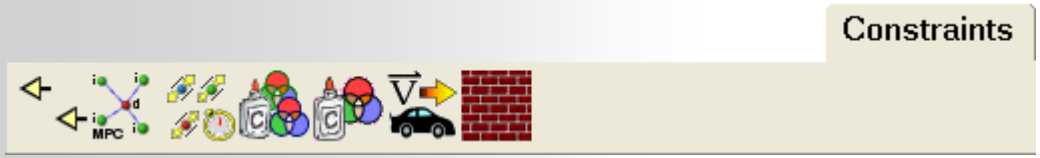
***Settings->Product* must also be set to an FEA version**

Properties



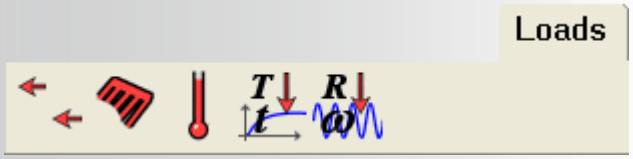
Create, read, write out material properties
Apply to geometry/elements

Constraints



Set constraints, displacements, define contacts, initial velocity, rigid walls

Loads



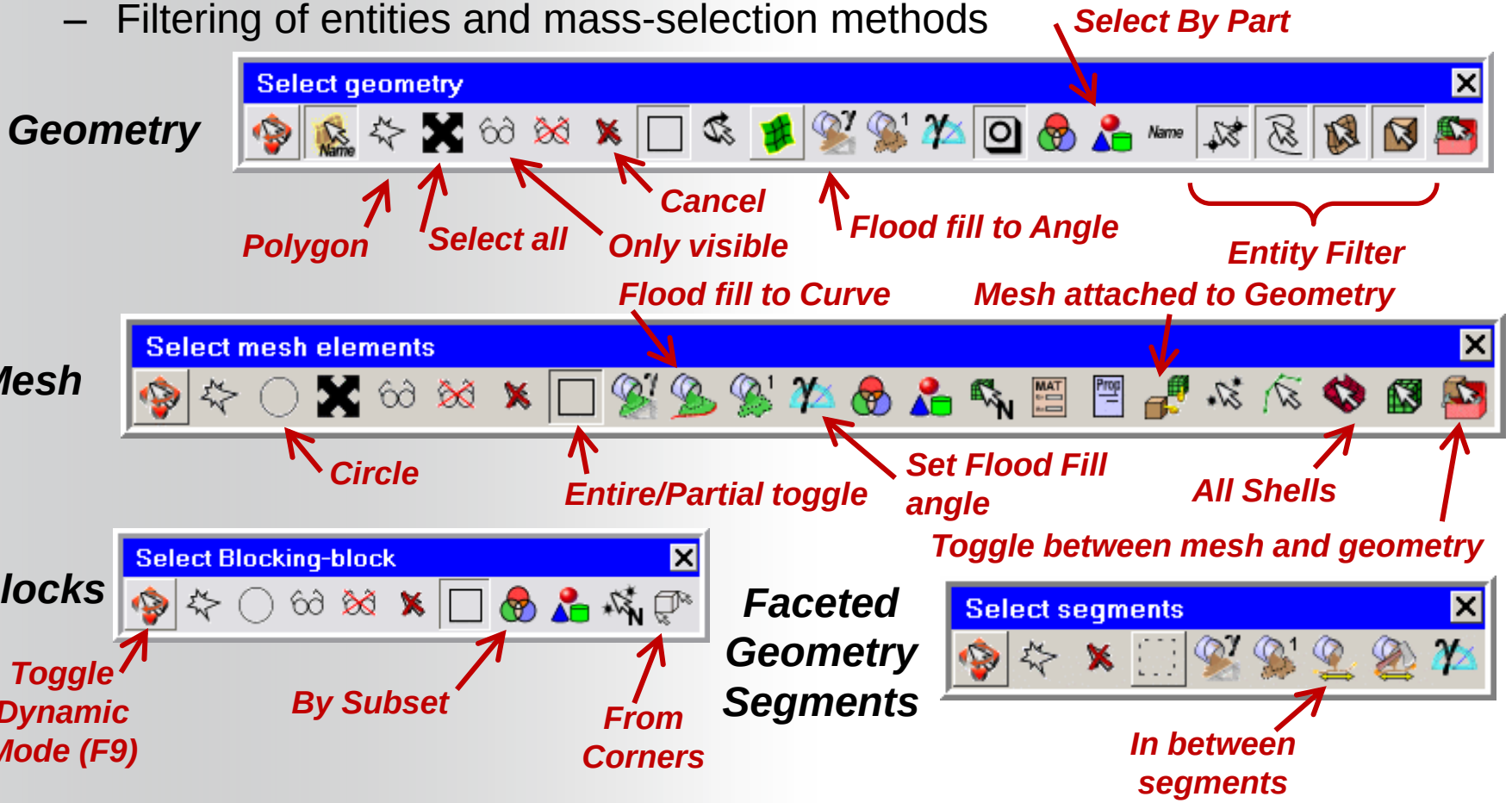
Set force, pressure and temperature loads

Solve options

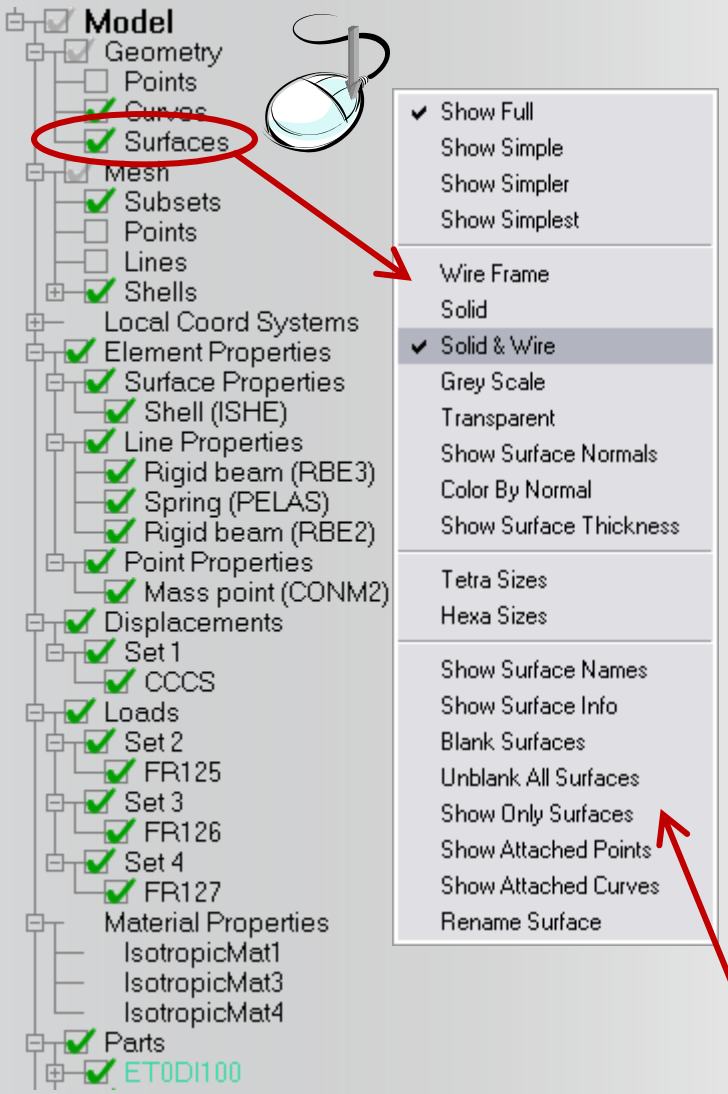
Set parameters, attributes, create subcases, write out input file, run solver

Selection Toolbar

- **During select mode, popup selection toolbar appears**
  - Some tools are common to all, others are contextual
  - Linked to select mode hotkeys
  - Filtering of entities and mass-selection methods



Model Tree



To toggle on/off various sections of the model

Main Categories are:

- Geometry, Mesh, Blocking, Parts
- Local Coord Systems, Element Properties, Connectors, Displacements, Loads and Material Properties

Toggle check boxes to blank/unblank

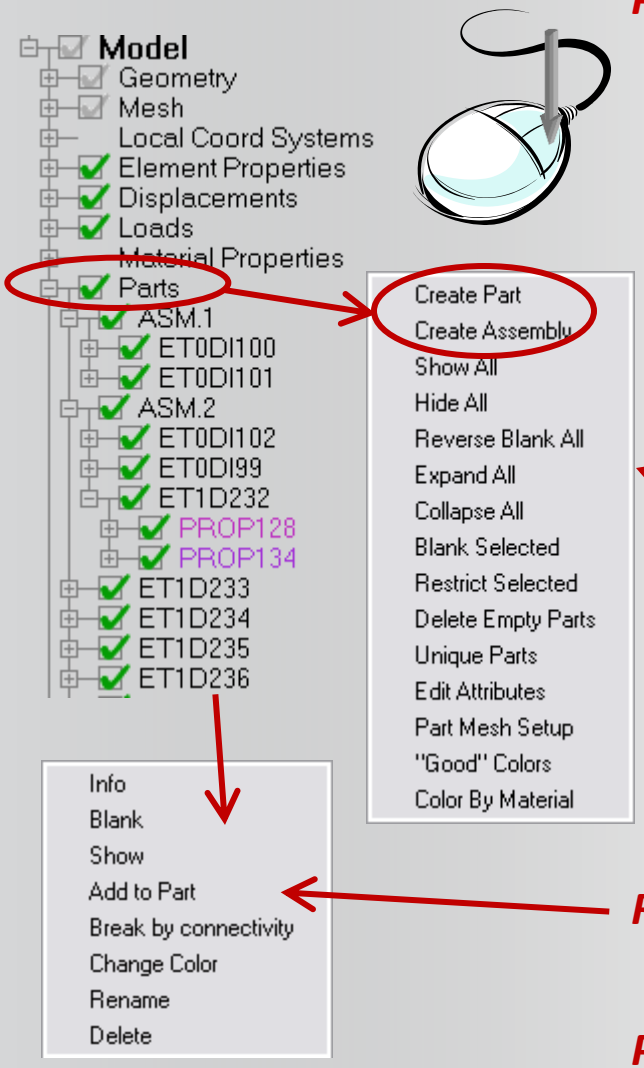
- Blanked/inactive
- Visible/active
- Partially visible/active: some sub members turned on, some turned off

Click on plus sign to expand tree

- Expose sub members

Right mouse click on each item for display options

Model Tree: Parts



Parts

- **Grouping of mesh, geometry, and blocking entities**
  - **Based on boundary condition/property**
  - **Based on mesh size (can set mesh size by part)**
  - **Based on material property**
  - **Just to partition large model**
- **Left click to blank/unblank all entities within part**
- **Color coded: Part name matches entity screen display**
- ***Right Mouse Button* on *Parts* to access:**
  - ***Create Part***
  - ***Create Assembly***
  - ***Delete Empty Parts***
  - ***Etc.***

***RMB* on specific part names allows options to modify or delete the individual parts**

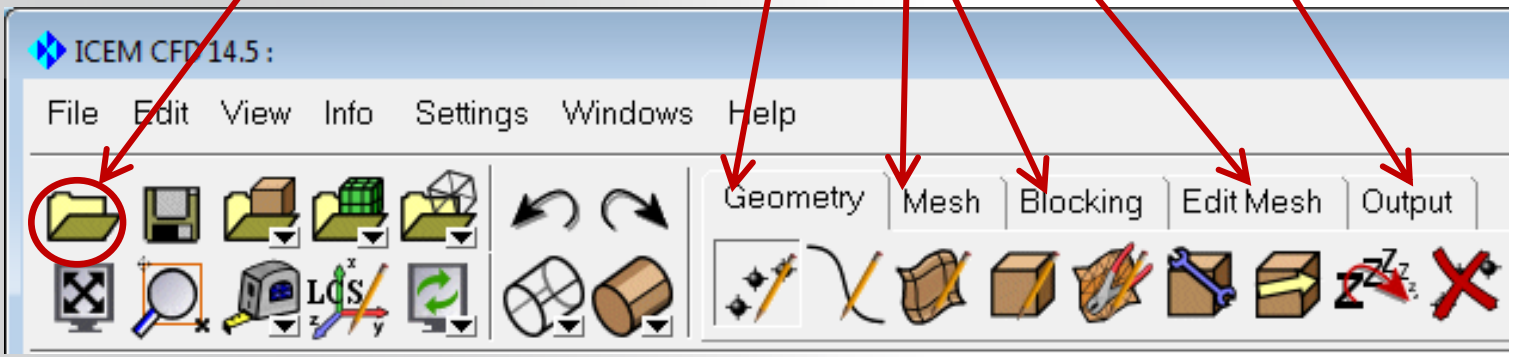
***Properties* are shown as a sub branch of the part**

- **Double Left Click or RMB > Modify to modify element properties**

Workflow

Typical ICEM CFD Workflow:

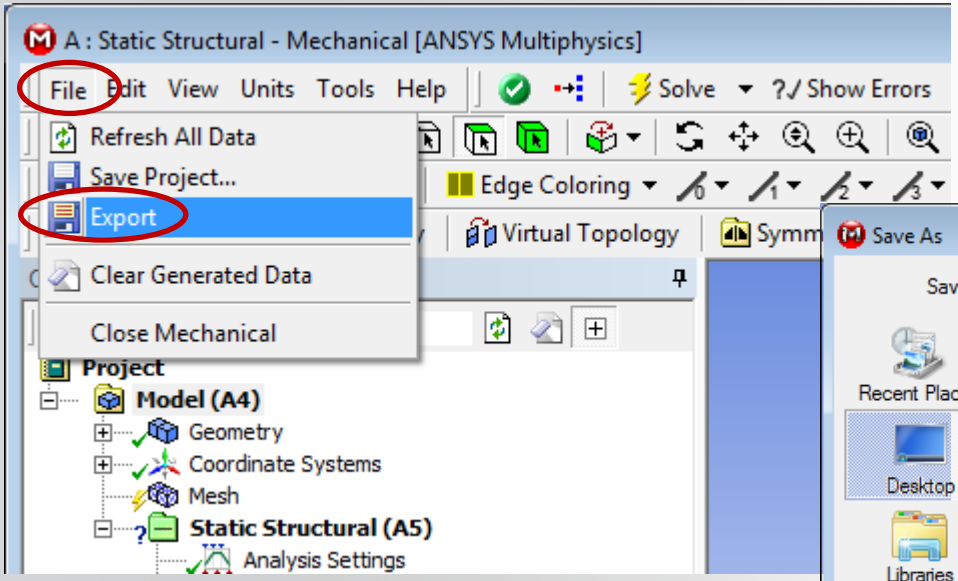
- **Create/open new project**
- **Import/Create geometry**
- **Build topology/Clean geometry/Create geometry**
- **Mesh model (Possibly Hex Blocking)**
- **Check/edit mesh**
- **Output to Solver**

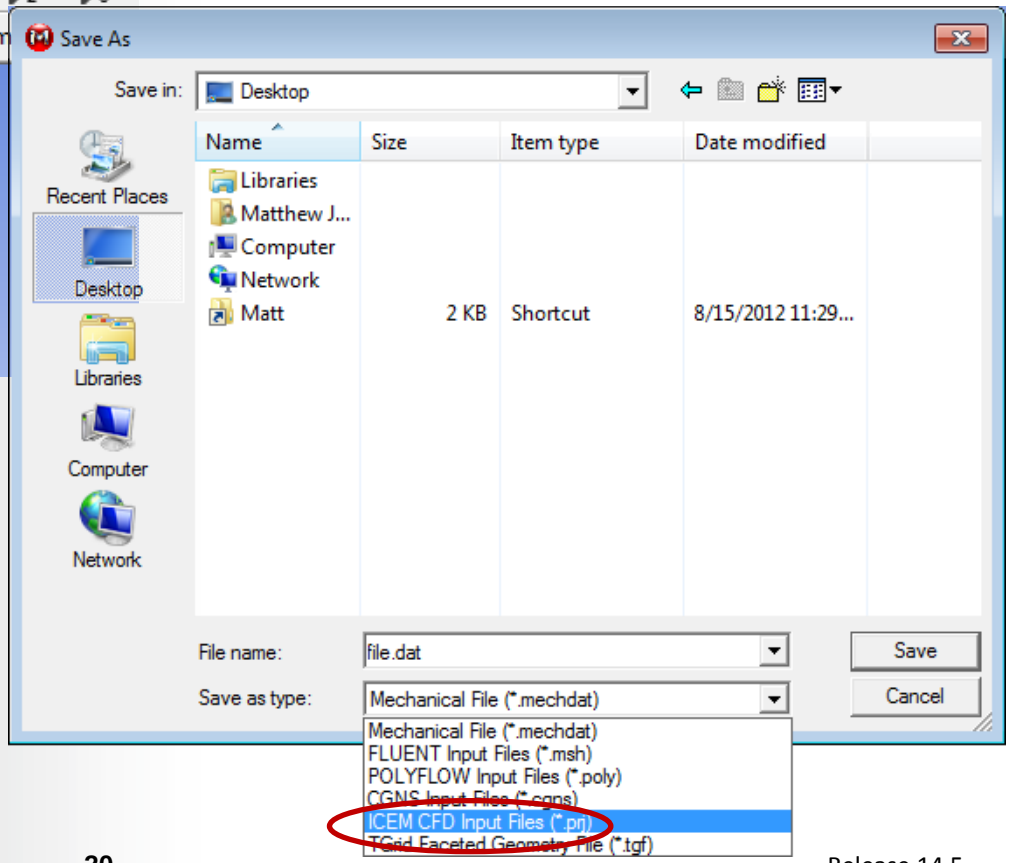


General Order of Workflow

Exporting ICEM CFD Files from Workbench

Export files from Mechanical Model (Simulation) or Meshing Application to open in ICEM CFD



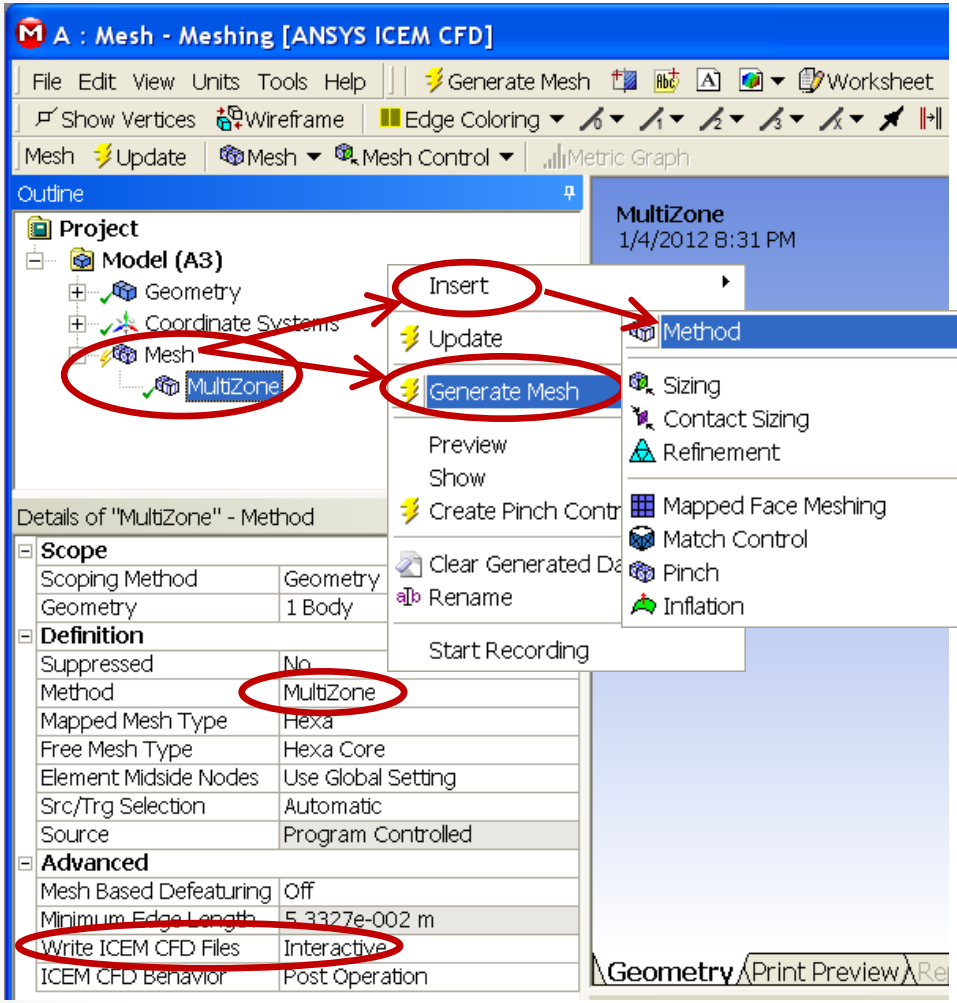


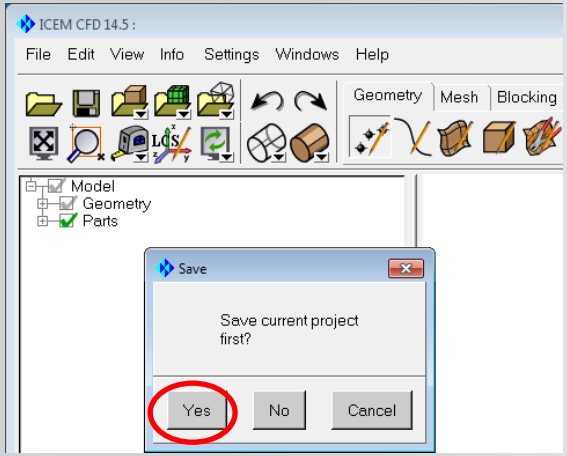
- **Some ICEM CFD capabilities have been integrated into the Meshing Application**
  - Tetra octree (patch independent)
  - 3D blocking fill (Multizone)
  - Autoblock (2D, uniform quad)
  - Body fitted cartesian

Workbench Interactive Link

Ansys ICEM CFD can be accessed from Workbench from certain mesh methods

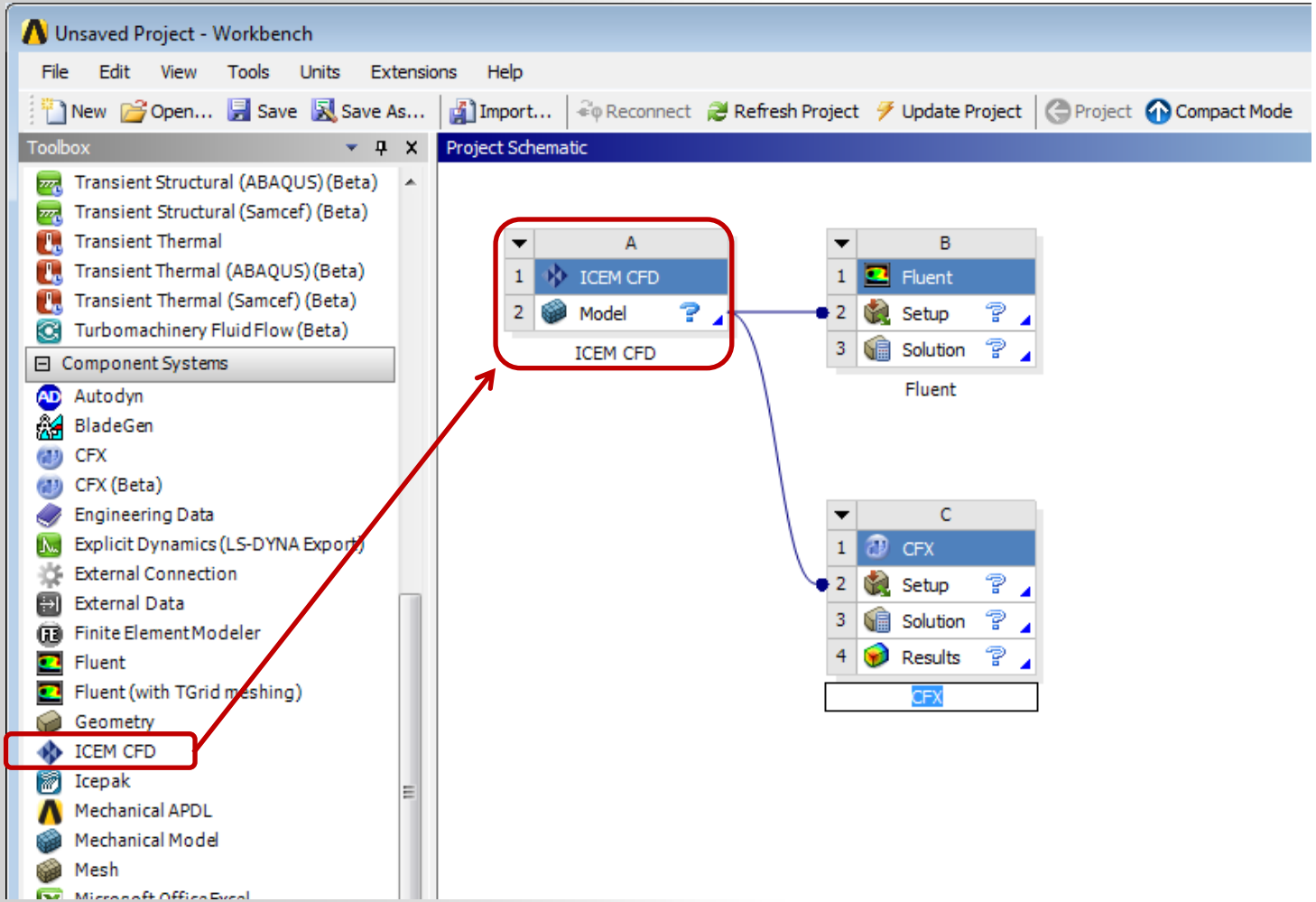
- **Insert a meshing method**
  - MultiZone
  - Patch Independent tetrahedrons
- **Set *Write ICEM CFD Files* to *Interactive***
- ***Generate mesh***
- **Edit or remesh within ICEM CFD, save project, then exit ICEM CFD**
  - Don't edit geometry in ICEM CFD





Workbench Project Schematic

Ansys ICEM CFD can be inserted into the Workbench project schematic from the Component Systems and linked with other systems



Let's Explore the GUI with a simple practice part

File > Import Geometry (parasolid or acis)

Show menu locations

Mouse controls

Change display options

- Solid/Wire, simple/full
- Views/Save view

Create Part

- Parts concept

Overview process

- Create/open/Save Project
- Build topology
- Define global and local mesh sizes
- Create volume mesh
- Edit/check quality, smooth
- Set up and write out to solver