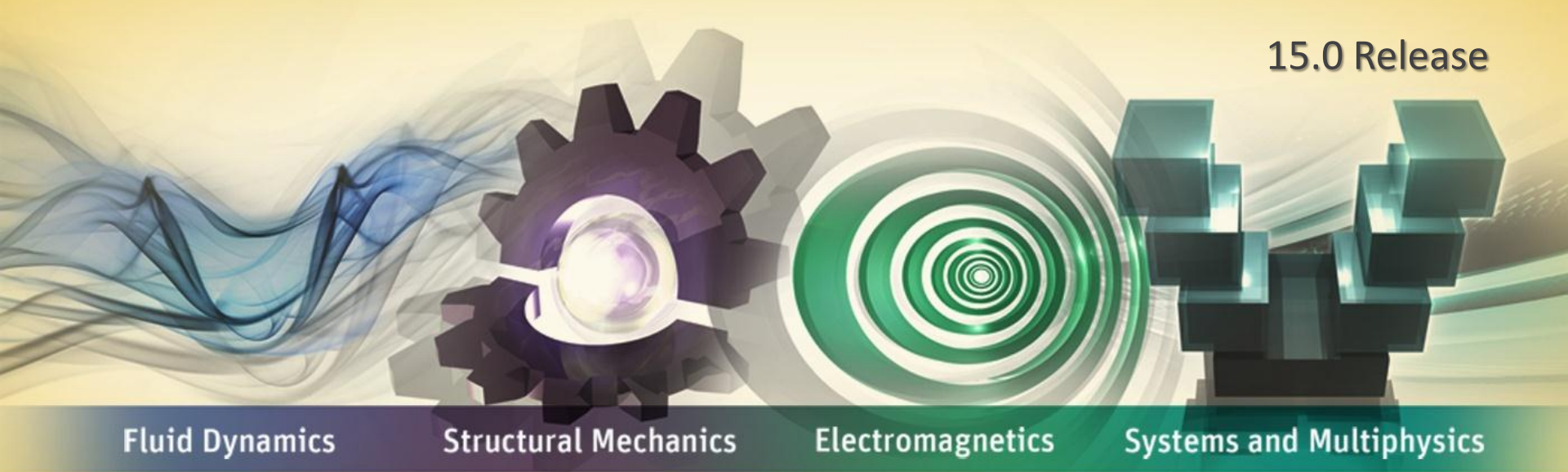


Lecture 1

Introduction to ANSYS ICEM CFD

15.0 Release

A horizontal banner with a yellow background. It features four main elements: a blue fluid flow visualization on the left, a purple gear with a glowing center, a green concentric circle pattern, and a stack of blue and black cubes on the right. Below these elements is a dark blue bar with white text labels for each section.

Fluid Dynamics

Structural Mechanics

Electromagnetics

Systems and Multiphysics

Introduction to ANSYS ICEM CFD

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

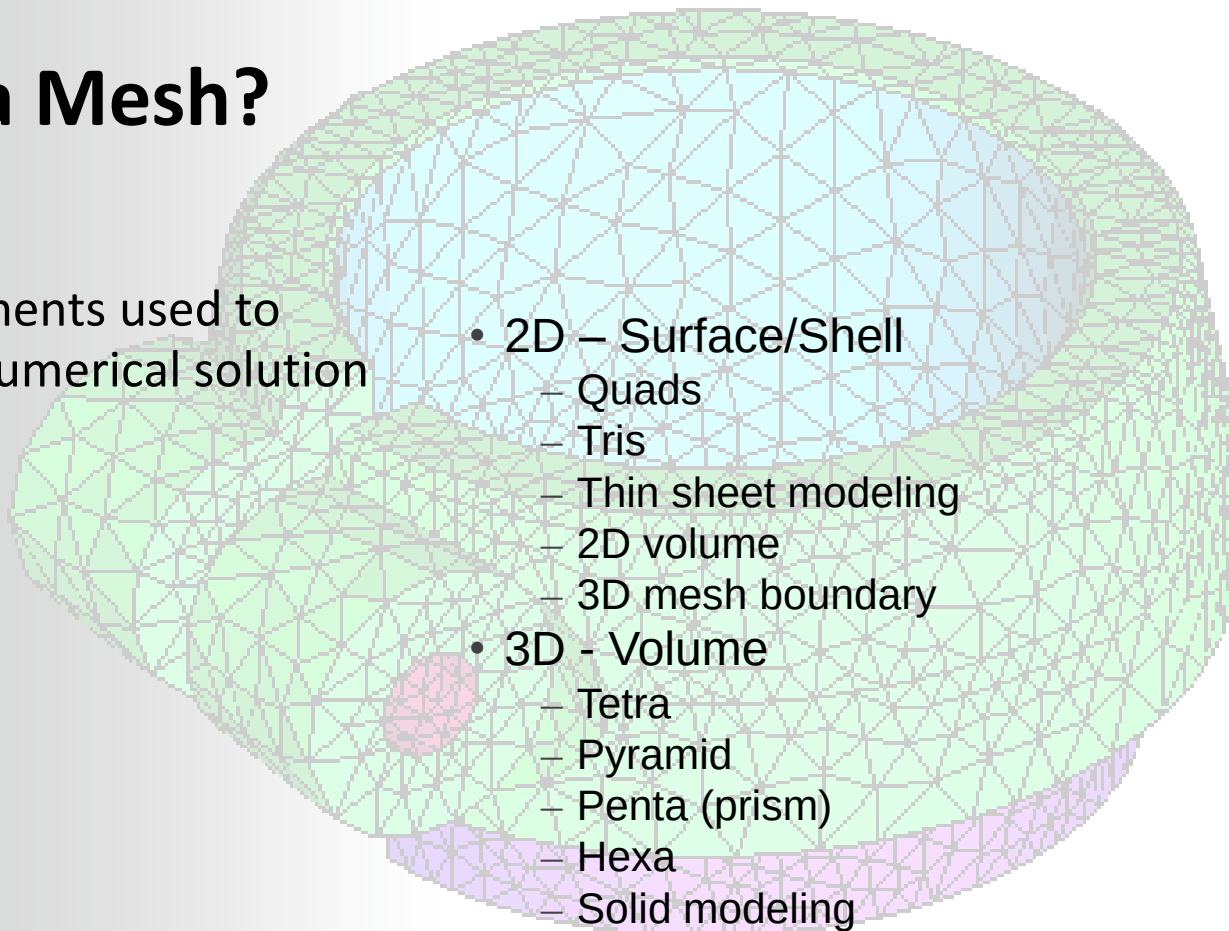
- Geometry import/editing
- Meshing (surface meshing, tetra, prism, hexa, hybrid meshes)
- Mesh Editing (smoothing, error checks, individual element editing)
- Solver setup

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

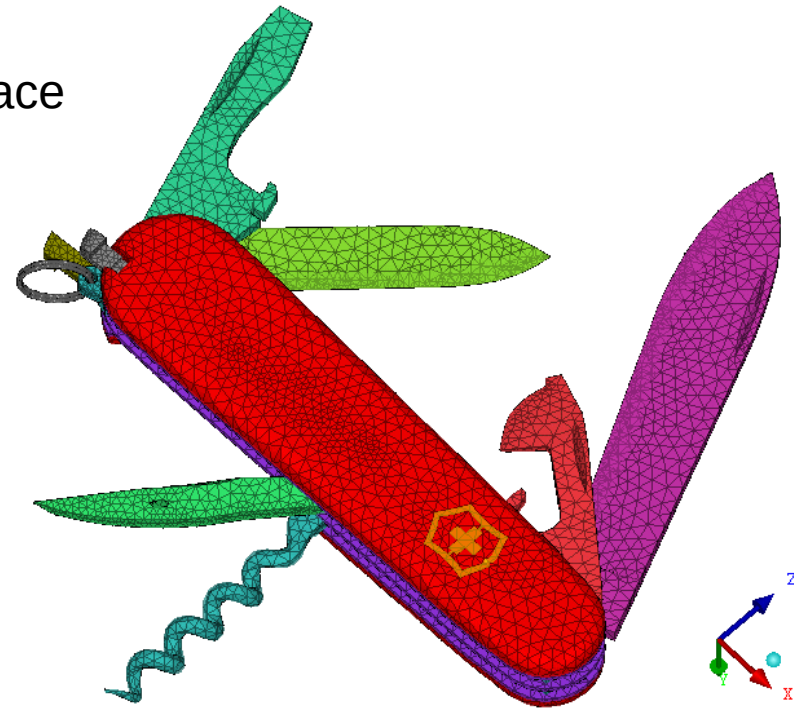
What is a Mesh?

Mesh

- Model 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



- **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 – common BC's**
- **Output to 100+ Solvers**
- **Scripting ... and much more...**



Utility
Icons

The screenshot displays the ANSYS R15.0 GUI interface. The top window is titled "ICEM CFD 15.0" and contains a "Utility Menu" with "File", "Edit", "View", "Info", "Settings", and "Help". Below this is a "Function Tabs" bar with "Geometry", "Mesh", "Blocking", "Edit Mesh", and "Output". A "Selection Toolbar" is positioned above the main 3D model area, with the label "Select mesh elements".

On the left side, the "Model Tree" lists the hierarchy: Geometry, Subsets, Points, Curves, Surfaces, Bodies, Mesh, and Parts. The "Parts" list includes BOLT_SIDE1, BOLT_SIDE2, CLAMP, GEOM, HOLE, and LEDGE. Below the Model Tree is the "Smooth Elements Globally" panel, which includes a "Quality" section with "Smoothing iterations" set to 5, "Up to value" set to 0.20, and "Criterion" set to Aspect ratio. The "Smooth Mesh Type" section has "TETRA_4" selected. The "Smooth Parts/Subsets" section has "Method" set to "All parts".

The main 3D model area shows a complex meshed part with a "Display Triad" (X, Y, Z axes) in the bottom right. Below the 3D model is the "Message Window" displaying quality metrics:

0.464649	->	0.498045	: 4769 (5.382%)
0.431253	->	0.464649	: 3863 (4.360%)
0.397857	->	0.431253	: 2068 (2.334%)
0.364461	->	0.397857	: 47 (0.053%)
0.331065	->	0.364461	: 5 (0.006%)

Below the message window is a "Histogram Window" showing a bar chart of quality metrics. The x-axis represents the quality metric (Aspect ratio) from 0.33 to 1.0, and the y-axis represents the count from 0 to 24. The histogram shows a distribution of quality metrics, with the highest frequency occurring between 0.4 and 0.5.

At the bottom of the GUI, there is a "Data Entry Panel" with a "Refresh Histogram" button and "Apply", "OK", and "Dismiss" buttons. The "Message Window" also has "Log", "Save", and "Clear" buttons.

File and Directory Structure

Use of many files

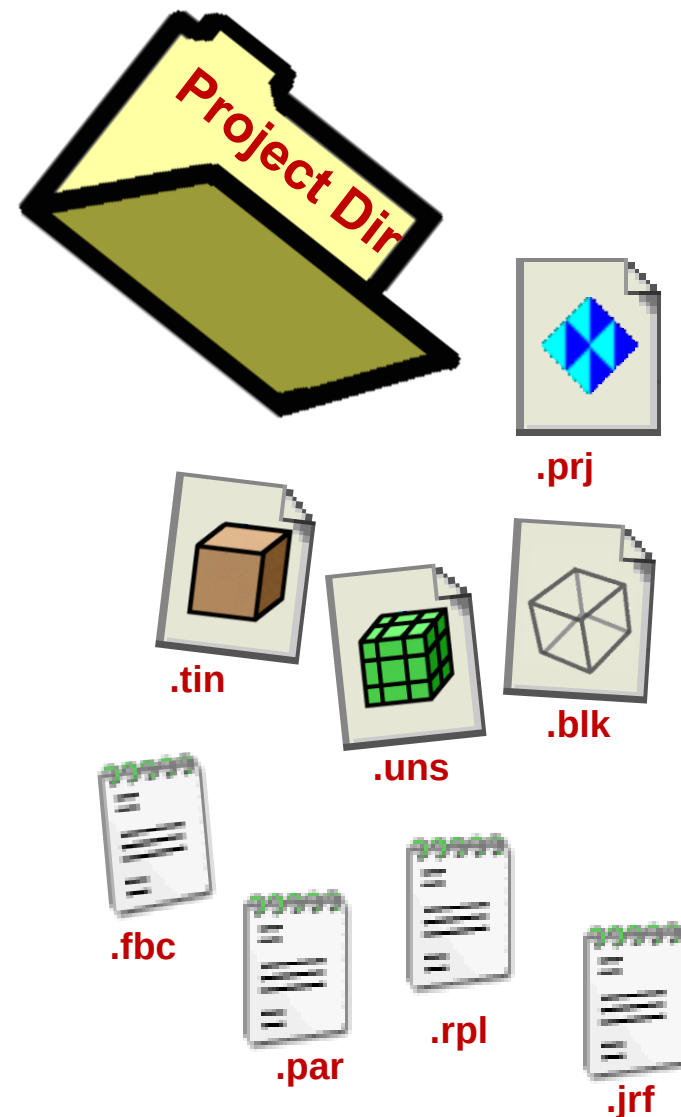
- Not one large common database
- For faster input/output

All files can optionally be associated within a **Project**

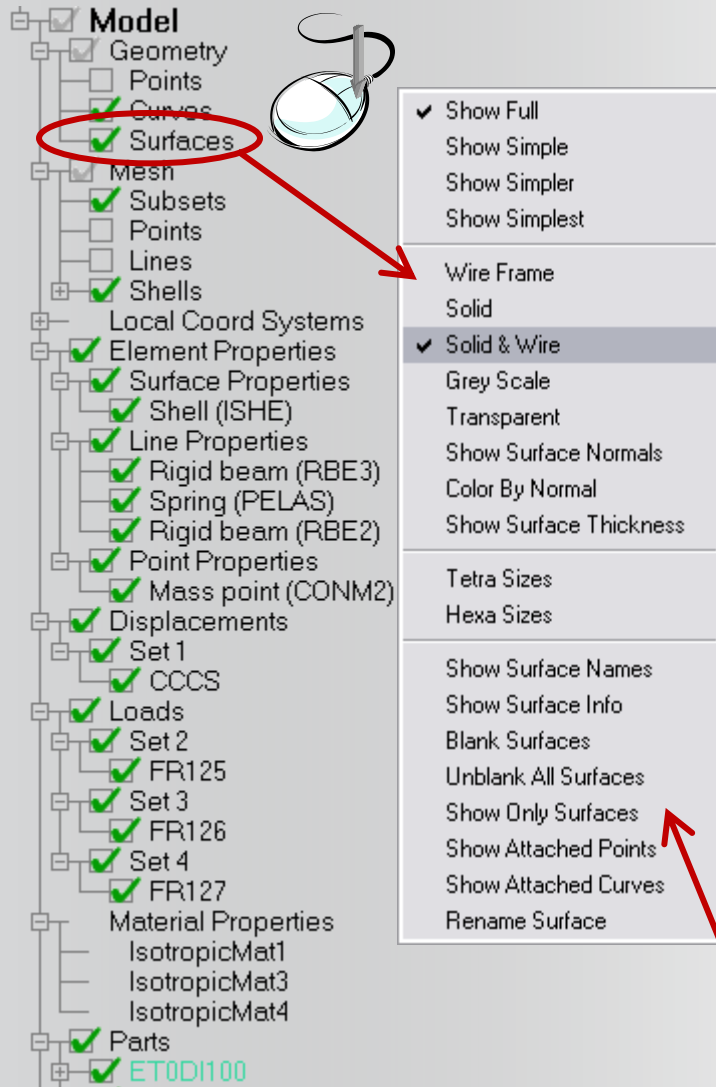
- Establishes working directory when opened or saved
- Settings (*.prj) file contains associated file names

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)



Model Tree



To toggle on/off various sections of the model

Main Categories are:

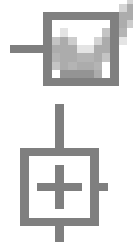
- Geometry, Mesh, Blocking, Parts
- Local Coord Systems, Element Properties, Connectors, Displacements, Loads and Material Properties

Toggle check boxes to blank/unblank

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

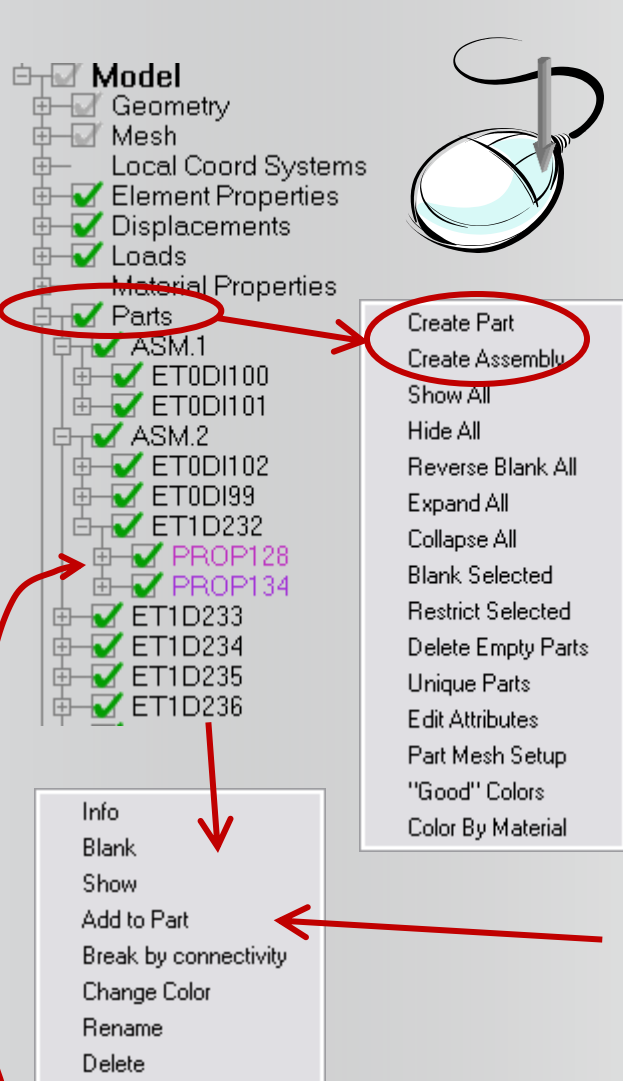
Click on plus sign to expand tree

- Expose sub members



Right mouse click on each item for display options

Model Tree: Parts



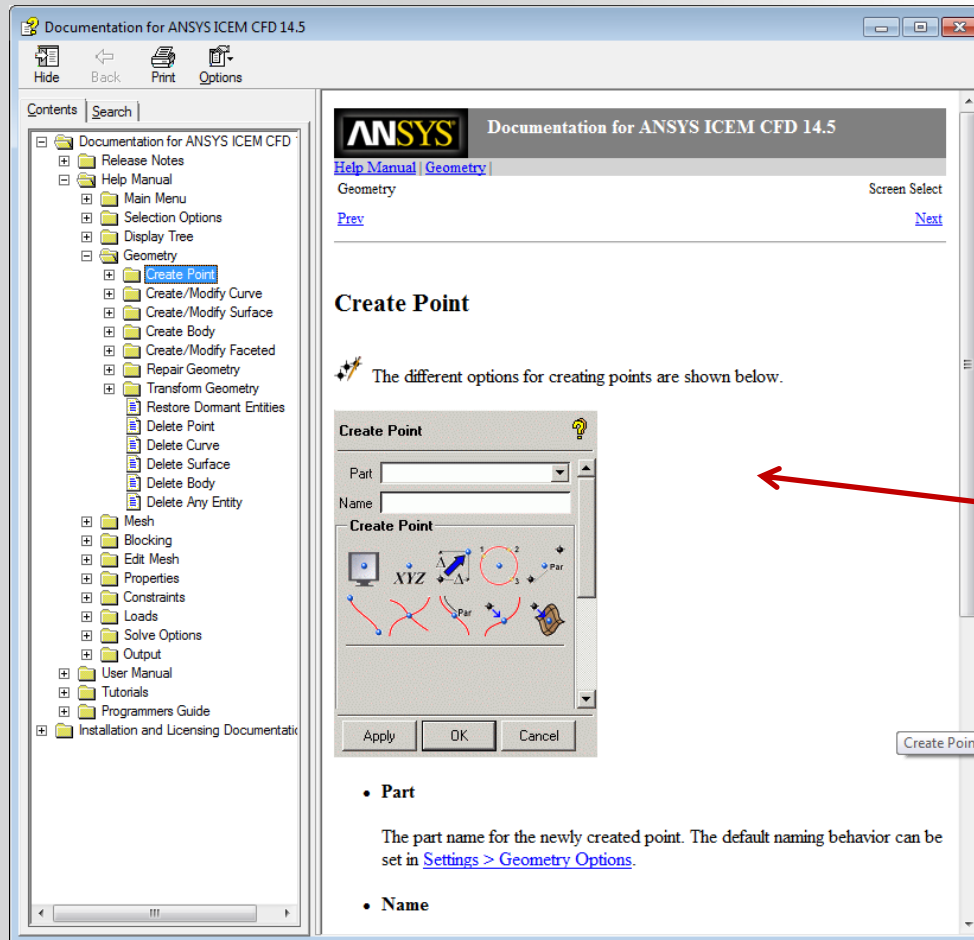
Parts

- Grouping of mesh, geometry, and blocking entities
 - Based on boundary condition/property
 - Based on mesh size (can set mesh size by part)
 - Based on functions that work by part selection
 - 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

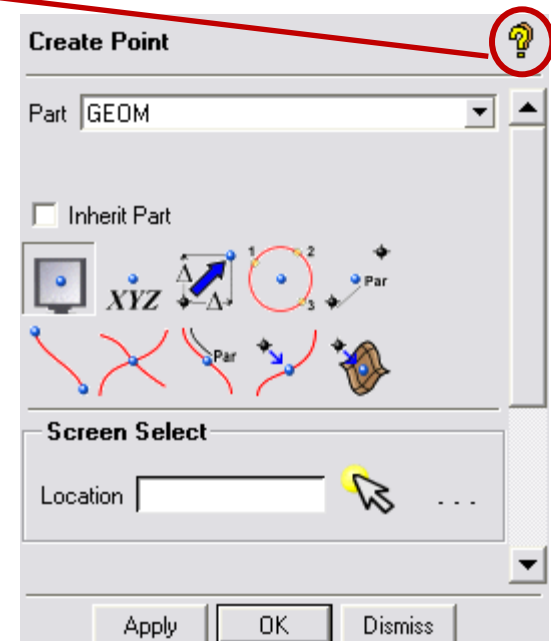
- **Double Click** or **RMB > Modify** to modify element properties



Menu Driven

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

Hyper-link to specific topic

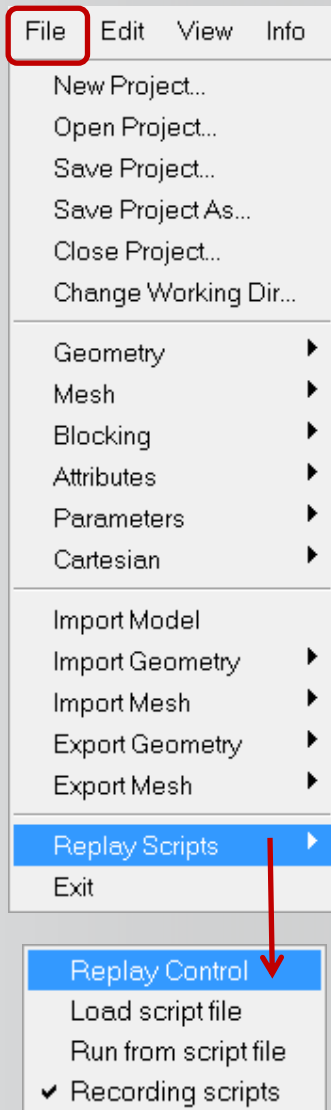


- Bubble explanation with cursor positioning



Scripting

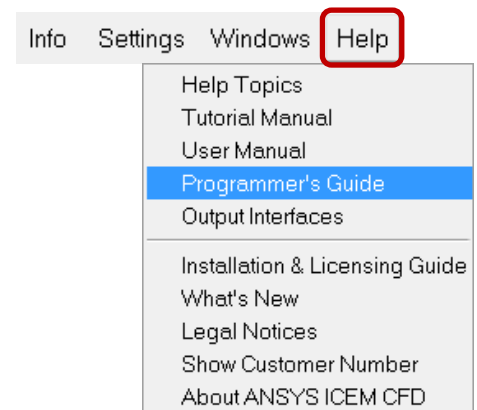
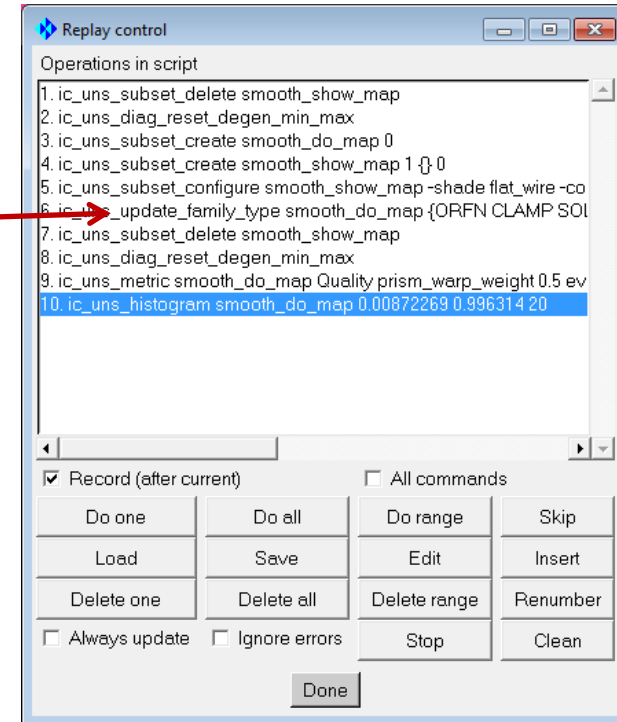
Scripts can be run and recorded from the File menu



- **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 ICEM CFD commands

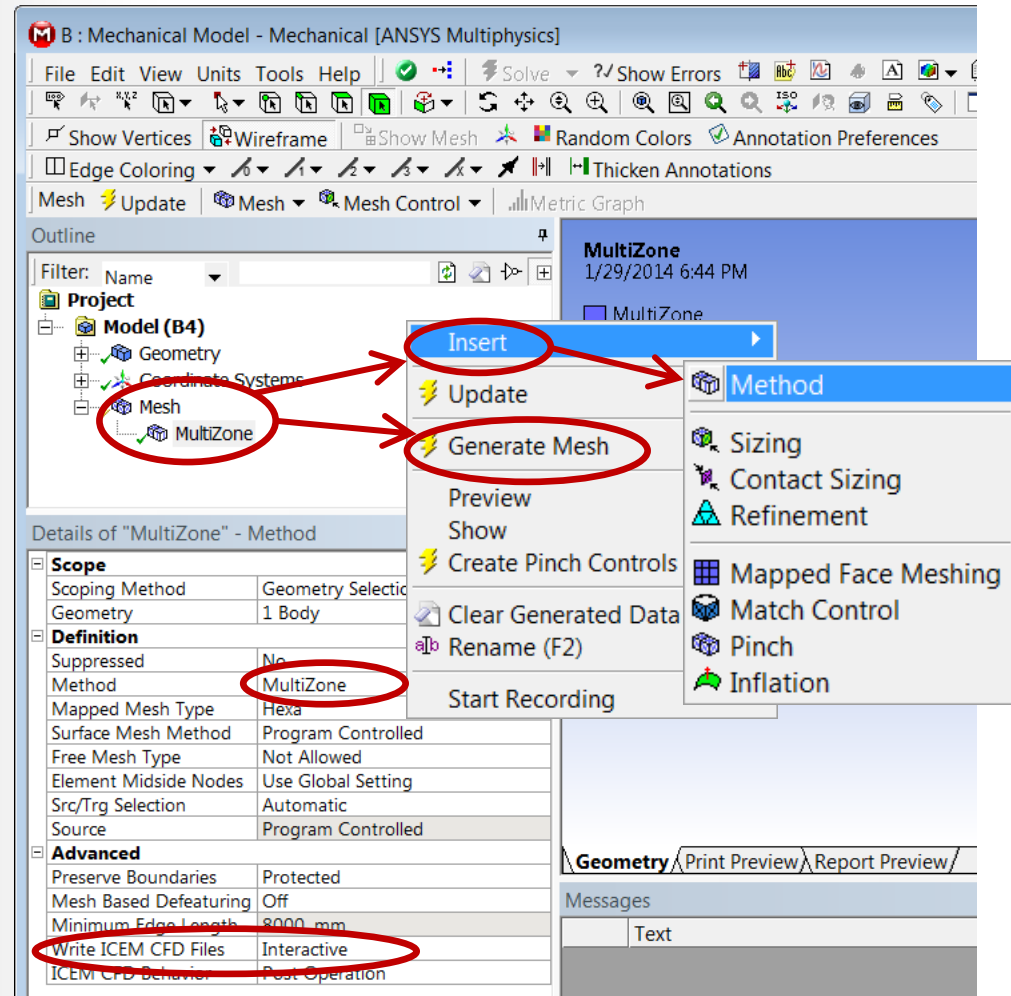
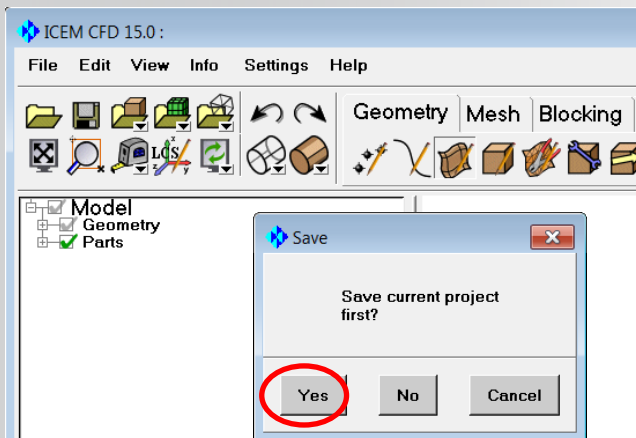
- The first section, **ICEM CFD Scripting Basics**, gives you a quick start to scripting with ICEM CFD



Workbench Interactive Link

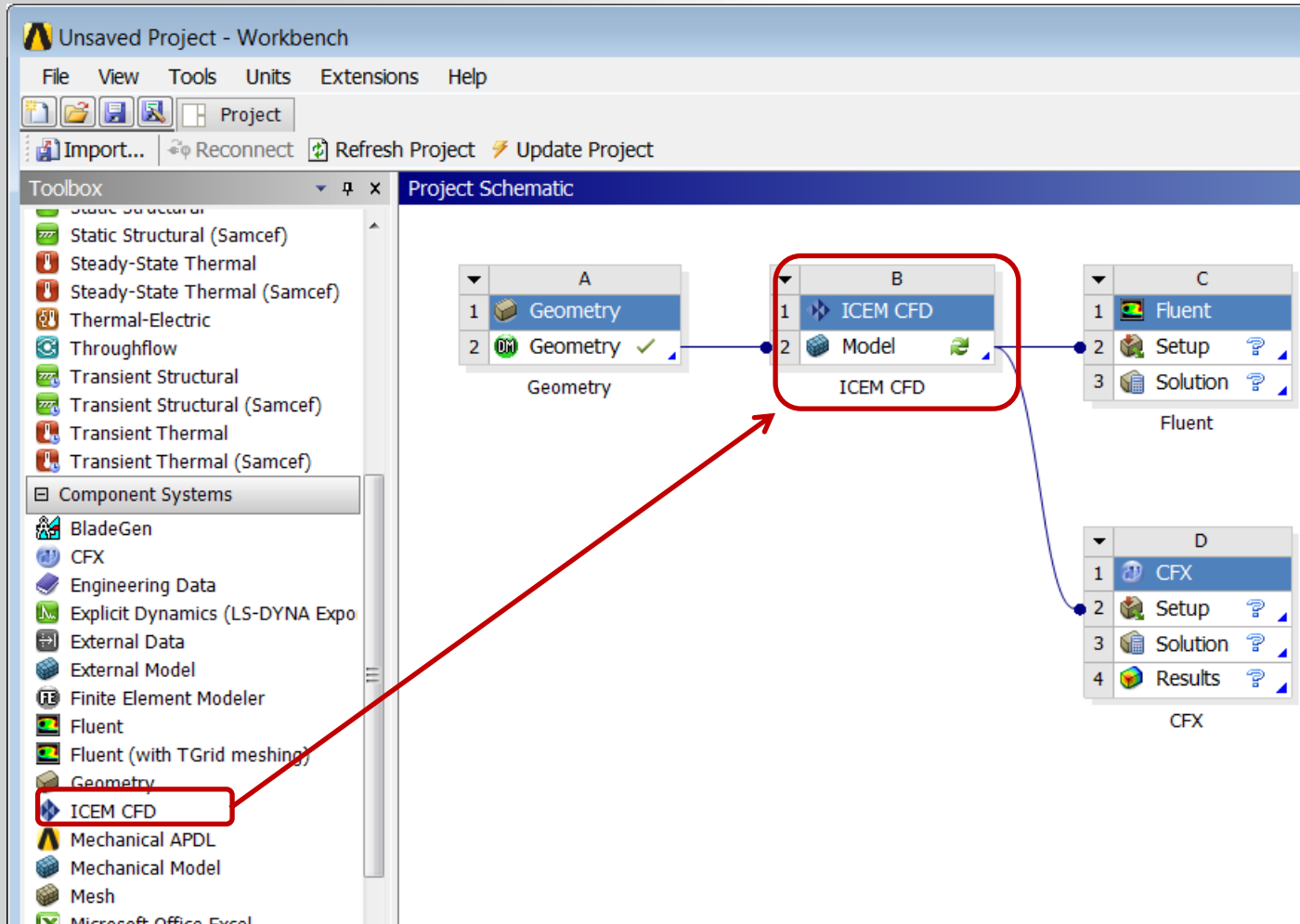
Ansys ICEM CFD can be accessed from Mechanical and Meshing Application

- 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



1. ANSYS Customer Portal

- <https://support.ansys.com>

Contains over 50,000 support assets powered by a modern web user interface and powerful search engine.



2. Call 1-800-711-7199

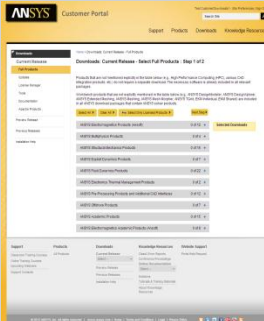
3. Email techsupp@ansys.com

- These last 2 methods will be routed through the dispatcher



Submit and review service requests

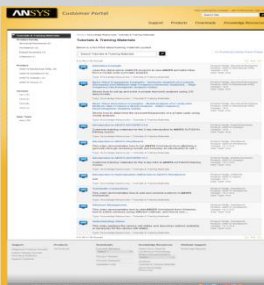
If you cannot find the answer to your question within the ANSYS Customer Portal then you can submit a service request. A member of ANSYS technical support will then get back to you with advice or a solution.



Download the latest software and updates

Download ISO images if you wish to create a DVD which is recommend for installations on multiple computers and allows you to keep an archive of the installation for later re-use.

Individual downloads can also be selected if you have a limited speed internet connection.



Download classroom and video training material

Training and tutorial material are available for both a broad range of ANSYS products and user's experience. Search the hundreds of courses available and improve your knowledge of ANSYS software.

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

