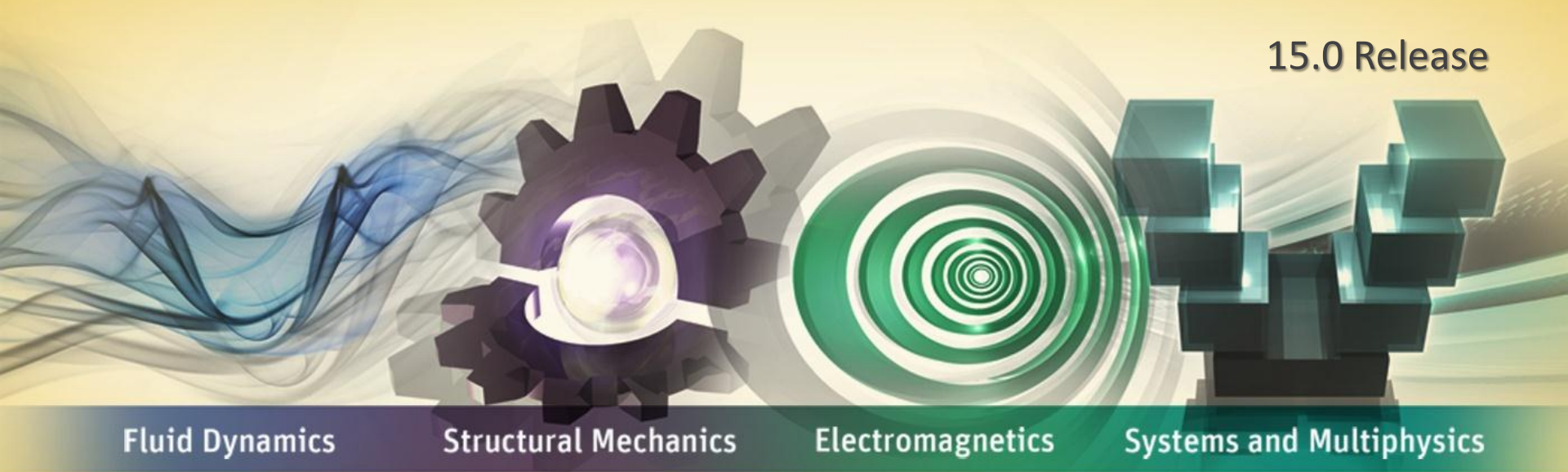


Lecture 7

Output to Solver

15.0 Release

A horizontal banner with a yellow-to-orange gradient background. It features four distinct 3D visualizations: blue fluid flow lines on the left, a dark purple gear with a glowing center, a green concentric circle pattern, and a stack of teal cubes on the right. Below these images is a dark teal bar containing four white text labels.

Fluid Dynamics

Structural Mechanics

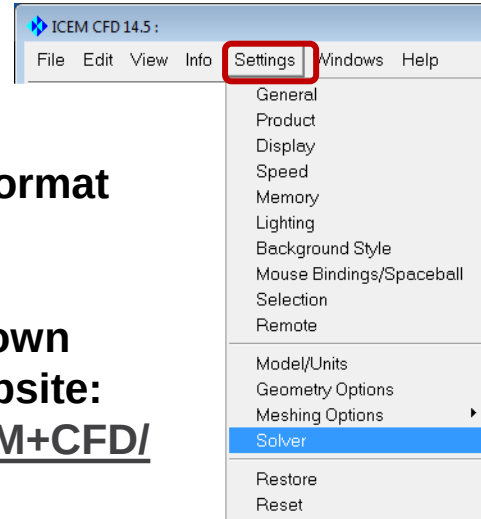
Electromagnetics

Systems and Multiphysics

Introduction to ANSYS

ICEM CFD

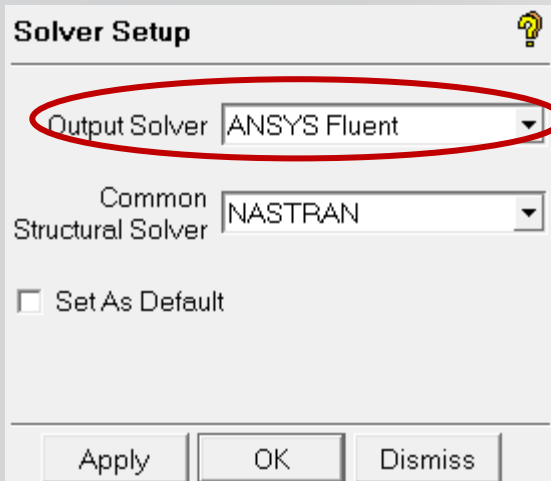
Output to Solver - Selecting Solver



Selecting solver

- Use the red toolbox in the **Output** tab to select the solver format
- This same menu is accessed with **Settings > Solver**
- 123 solver formats available
- All output formats can be set with the **Output Solver** pulldown
- Help on each solver format can be found on the Ansys website:
<http://www.ansys.com/Products/Other+Products/ANSYS+ICEM+CFD/Output+Interfaces/Output+Interfaces+TOC>

- Also found in **Help > Output Interfaces**



ABAQUS	ACE-U	AcFlux	ACRi	ACUSOLVE	VULCAN	SCRYU
ADINA	AIRFLO3D	ALPHA-FLOW	ANSYS	ATTILA	WIND	STARCD
AUTOCFD	BAGGER	CEDRE	CFD-ACE	CFDesign	WINDMASTER	TDF
CFD++	CFL3D	CFX-4	CFX-5	CFX-TASCflow	ZEN	SC/Tetra
CGNS	CHAD	C-MOLD	COBALT	COMCO	USM3D	STARS
CONCERT3D	CRSOL	CRUNCH	CSP	DATEX	USMKV3V	TGRID
DSMC-SANDIA	DTF	EM	EXODUS	FANSC	VECTIS	SpecElem
FASTEST-3D	FASTU	FENFLOSS	FIDAP	FIRE	VRML	STL
FLEX	FLOTRAN	FLOWCART	FLOW-LOGIC	FLUENT V4	VSAERO/USAERO	TLNS3Dmb
FLUENT V6	GASP	GLS3D(ADH)	GMTEC	GSMAC-DF	Trio_U	SPECTRUM-CENTRIC
GUST	HAWK	HDF	IBM-BEM	ICAT	TSAR	TEAM
ICU	IDEAS	IMPNS	INCA	IPLES	UGRID	TRANAIR
KIVA-3	KIVA-4	LAURA	LL-DYNA3D	LS-DYNA3D	UH3D	POLY3D
MACS	MAGREC	MAZe	MOUSE	MULTIBLOCK	USA	POPINDA
N3SNATUR	NASTRAN	NEKTON	NOPO	NPARC	SAUNA	PRECISE
NSU3D	NS3D	NUMECA	PAB3D	PARC	SPLITFLOW	RADIOSS
PATRAN	PHOENICS	PLOT3D	PMARC	POLYFLOW	TNO	RTT

Mesh Formats

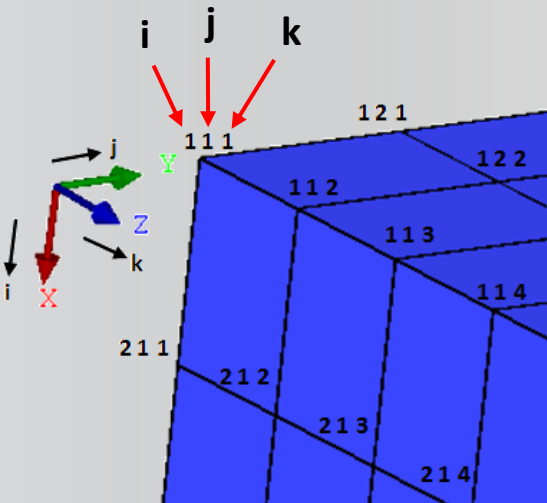
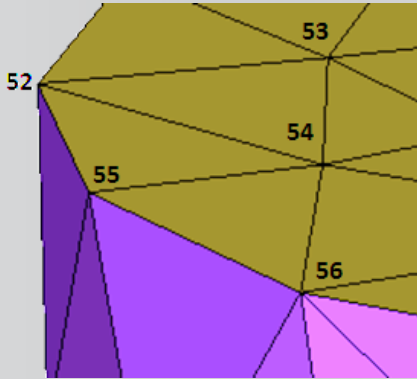
- There are 2 types of mesh formats that solvers read

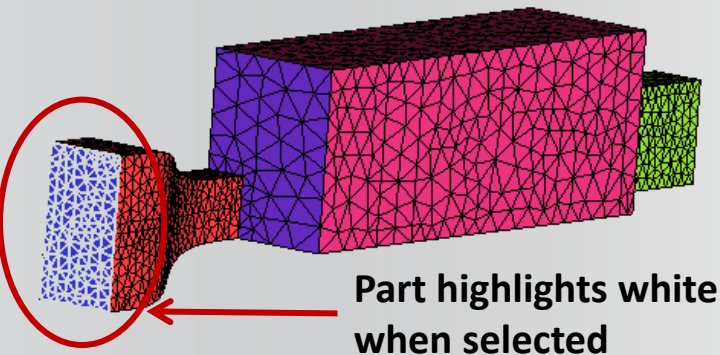
- **Unstructured**

- Most solvers use unstructured formats
- Some examples are **Fluent, Ansys CFX, CFD++, Abaqus, Ansys**
- Nodes have an ID and location
- The ICEM CFD unstructured mesh has a ***.uns** extension and can be made from any mesher in ICEM CFD

- **Multiblock structured**

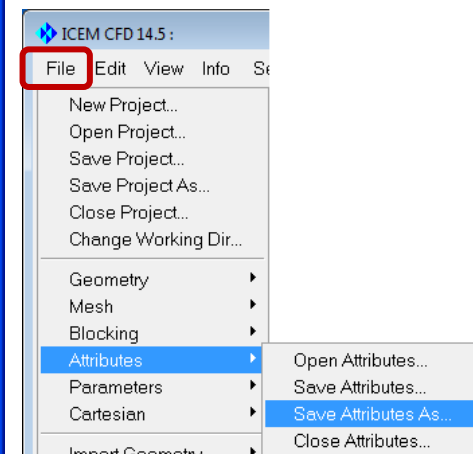
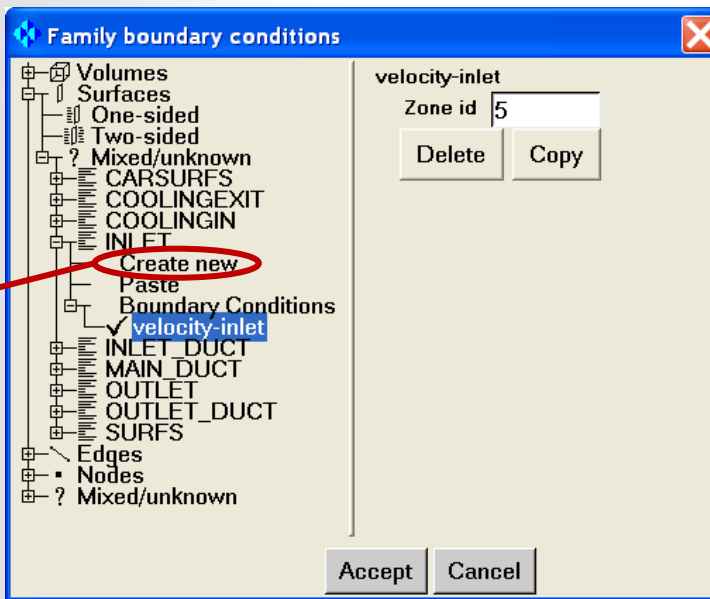
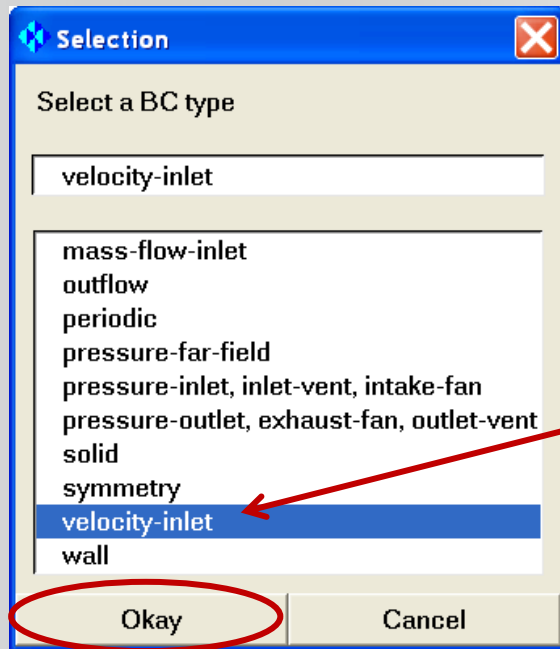
- Some older CFD solvers require this format
- Examples are **Plot3D, CFX-TASCFLOW, KIVA-3V**
 - **CGNS** supports both unstructured and multiblock structured
- Nodes have IJK index designation and location
- The ICEM CFD structured mesh has many files:
 - **Project.domain.1, project.domain.2, project.domain.3,** etc... for each mesh domain
 - A topology file, **topo_mulcad_out.top**, that describes how each domain is connected to the other domain.
 - A dummy file, **Project.multiblock**, that enables selection of all the multiblock mesh files of that project name by selecting this one file
- Only an ICEM CFD hexa blocking is capable of being written out in multiblock structured format



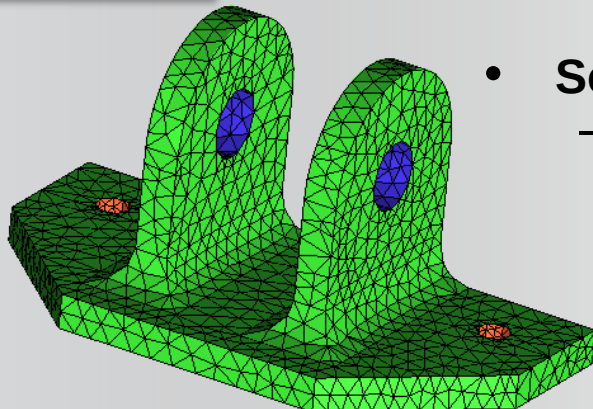


Part highlights white when selected

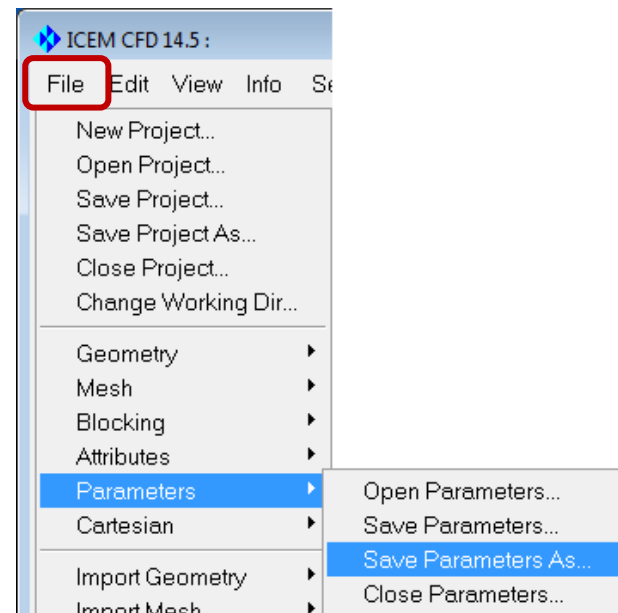
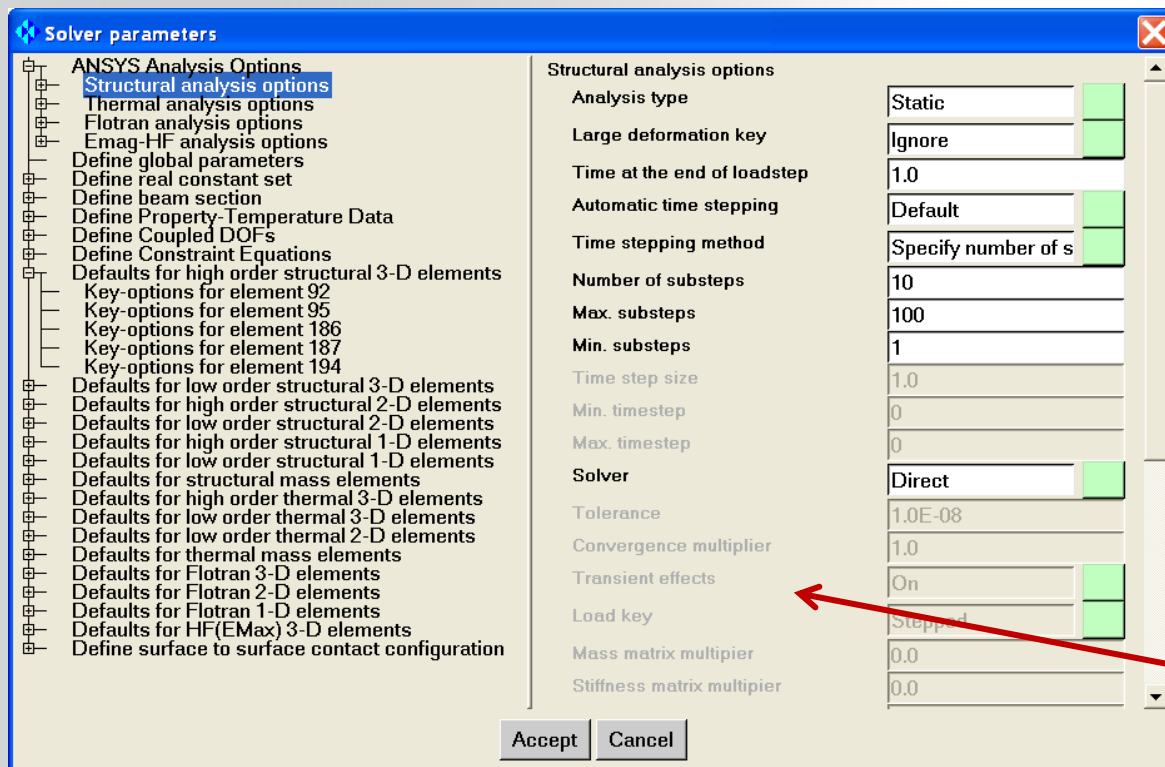
BC's shown for solver ANSYS Fluent



- Both CFD and FEA solvers allow boundary conditions to be set
 - Output > Boundary Conditions**
 - BC's are set on mesh elements grouped by part names
 - Tree structure organizes part names by dimension (2D, 3D, etc) of geometry and mesh in the part
 - Any part containing mixed dimensions (ex. curves [1D] and tri elements [2D]) will be grouped into **mixed/unknown**
 - Use **File > Attributes > Save Attributes As...** to save the BC file (***.fbc** and ***.atr** extensions)

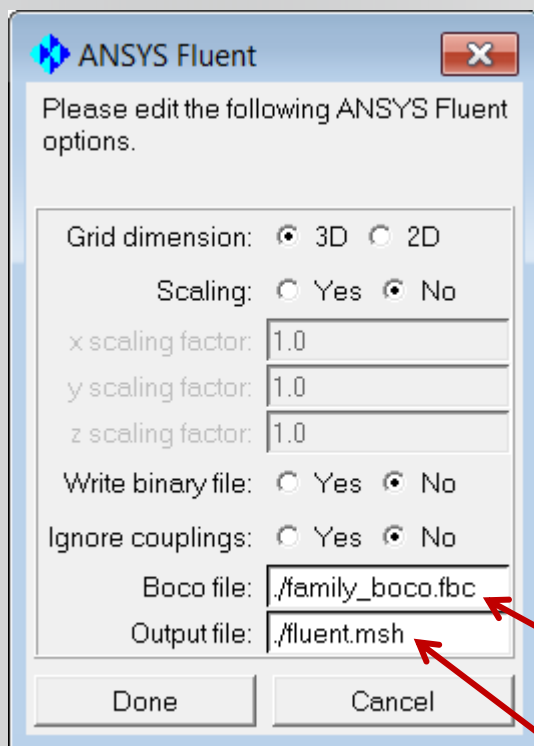
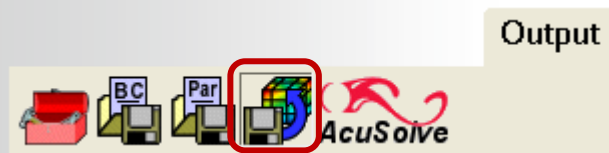


- Some structural solvers have global parameters
 - **Output > Edit Parameters**
 - If the solver type requires global parameters, you must bring up this menu before writing out and press **Accept**, even if not changing any parameters
 - Then use **File > Parameters > Save Parameters As...** to save the file (*.par extension)



Example parameters
shown for ANSYS solver

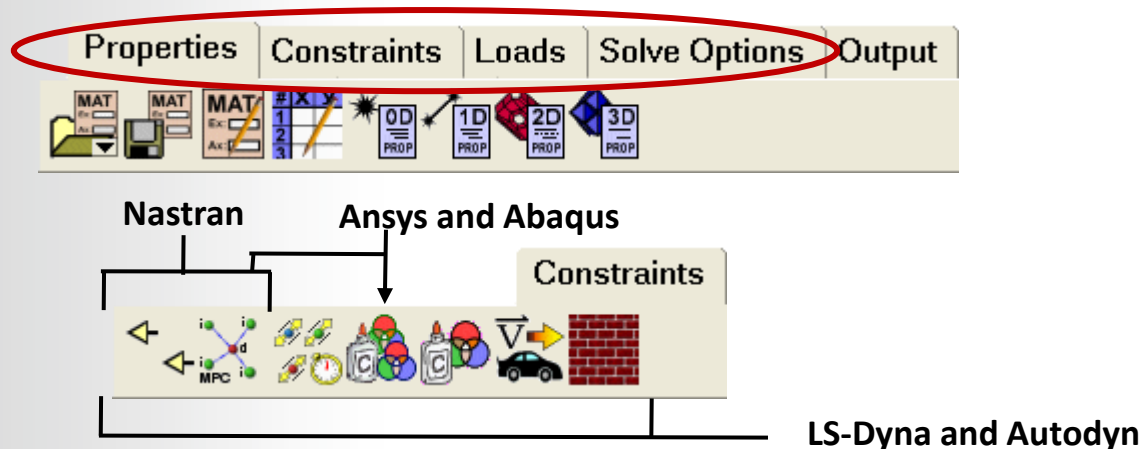
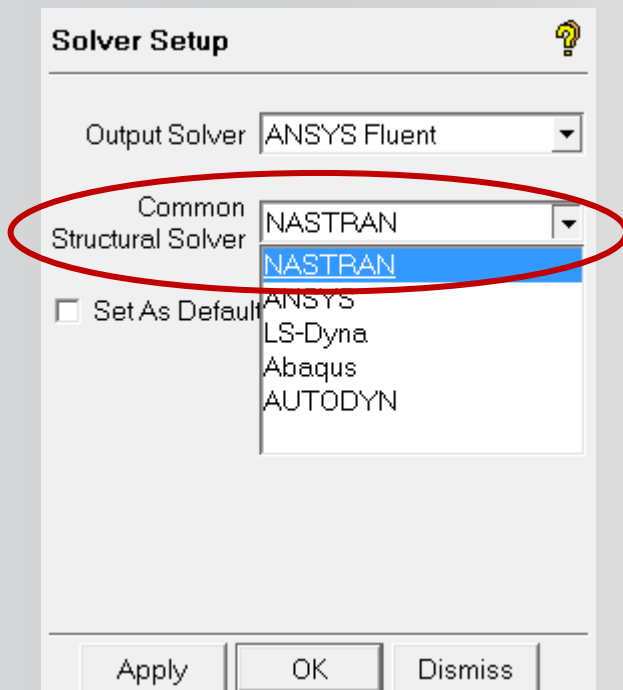
Write Input



- **Output > Write Input** to write the solver file
 - It will ask you to save a boundary condition file (***.fbc**) and the project
 - It's always safe to save these when asked, but not necessary if you know you saved them and didn't make any changes since then
 - It will then ask for the ICEM CFD mesh file to translate
 - ***.uns** file if the solver requires an unstructured mesh
 - ***.multiblock** file if the solver requires a structured mesh
 - A final menu will pop up that has specific options for the solver translation being used
 - The example on the left is for **ANSYS Fluent**
 - All translators will require a boundary condition file even if no BC's are set
 - A empty BC file will contain just the part names
 - There will always be a name for the output file

Special Structural Solvers

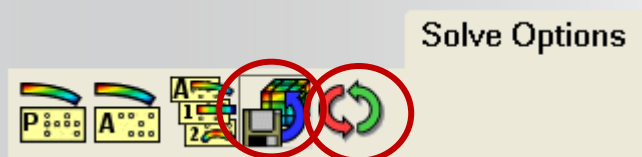
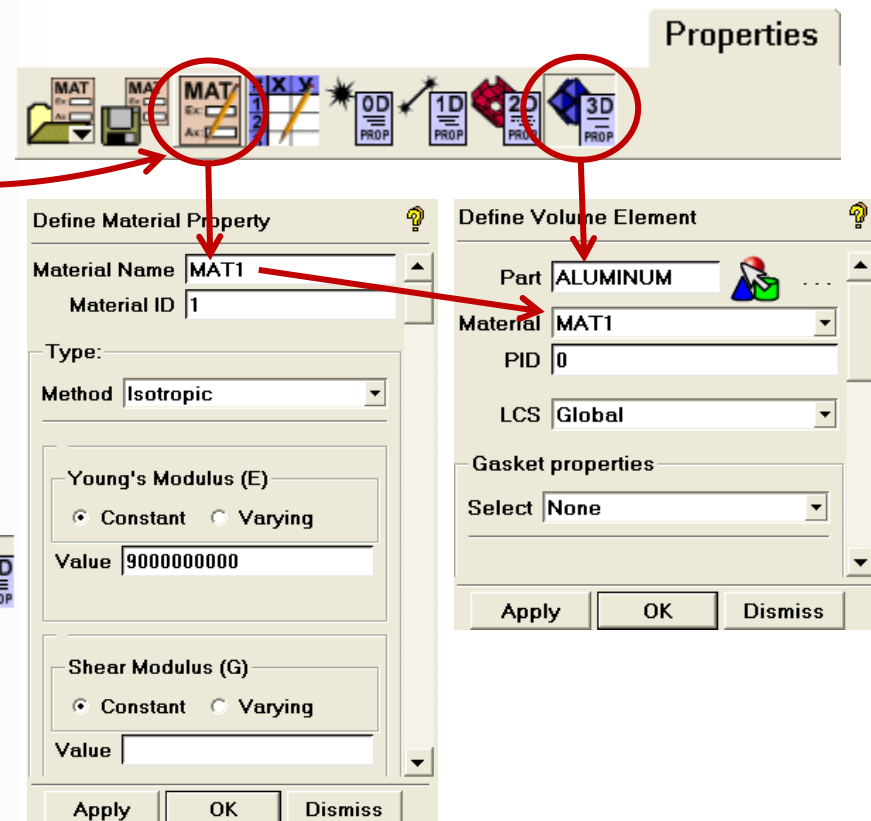
- 5 Special structural solvers
 - There are 5 structural solvers that have extended boundary condition setup – called loads and constraints – which are set using the **Common Structural Solver** dropdown
 - These are **Nastran**, **Ansys**, **LS-Dyna**, **Abaqus**, and **Autodyn**
 - These are set using the 4 extra tabs; **Properties**, **Constraints**, **Loads**, and **Solve Options**



- Constraints tab
 - All functions are only available for **LS-Dyna** and **Autodyn**
 - “**Define contact**” is allowable with **Ansys** and **Abaqus**
 - **Nastran** only allows the first two functions

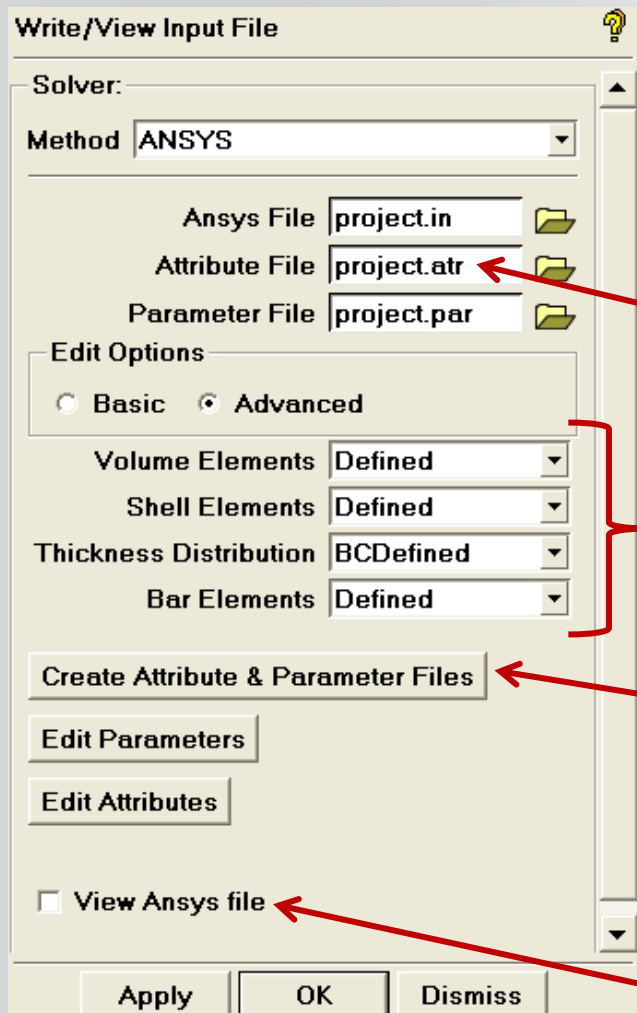
Special Structural Solvers - Procedure

- Define which elements to write out
 - First define a **material** in the **Properties** tab
 - Then define an element property on the element types and parts you want to write out
 - 3D property for volume elements
 - 2D property on shells
 - 1D property on line elements
 - 0D property on node elements
- Define any **Loads** and **Constraints** in the other tabs (optional)
- Use the **Solve Options** tab to write the mesh to the solver
 - Solve Options > Write/View Input File**



- Can submit solver run if the corresponding environment variable is set to find the solver executable
 - For Ansys, it is `ANSYS_EXEC_PATH`

Special Structural Solvers – Write Input



- If using the **Properties**, **Loads**, and **Constraints** tabs for one of the 5 special structural solvers, then use the **Solve Options** tab to write the file, not the **Output** tab
 - The **attribute** file (*.atr) is the BC file used with these special structural solvers
 - The **attribute** file is replaced with the boundary condition file (*.fbc) when using the **Output** tab
 - Any elements set to **All** or **None** will override any properties set in the **Properties** tab if not set to **Defined**
 - To communicate with the **Output** tab, click on **Advanced**, and then click **Create Attribute & Parameter Files**
 - You can then **Edit Parameters** and **Edit Attributes** here or in the Output tab
 - The attributes will then be the same as the boundary conditions
 - View input file here or in your own text editor

Workshop 7.1

Mesh Repair and Export

- Read geometry and create mesh
- Check mesh quality
- Identify areas of poor mesh
- Repair mesh
- Write out mesh to CFD Solver

