Ennova Technologies

Training - Ennova

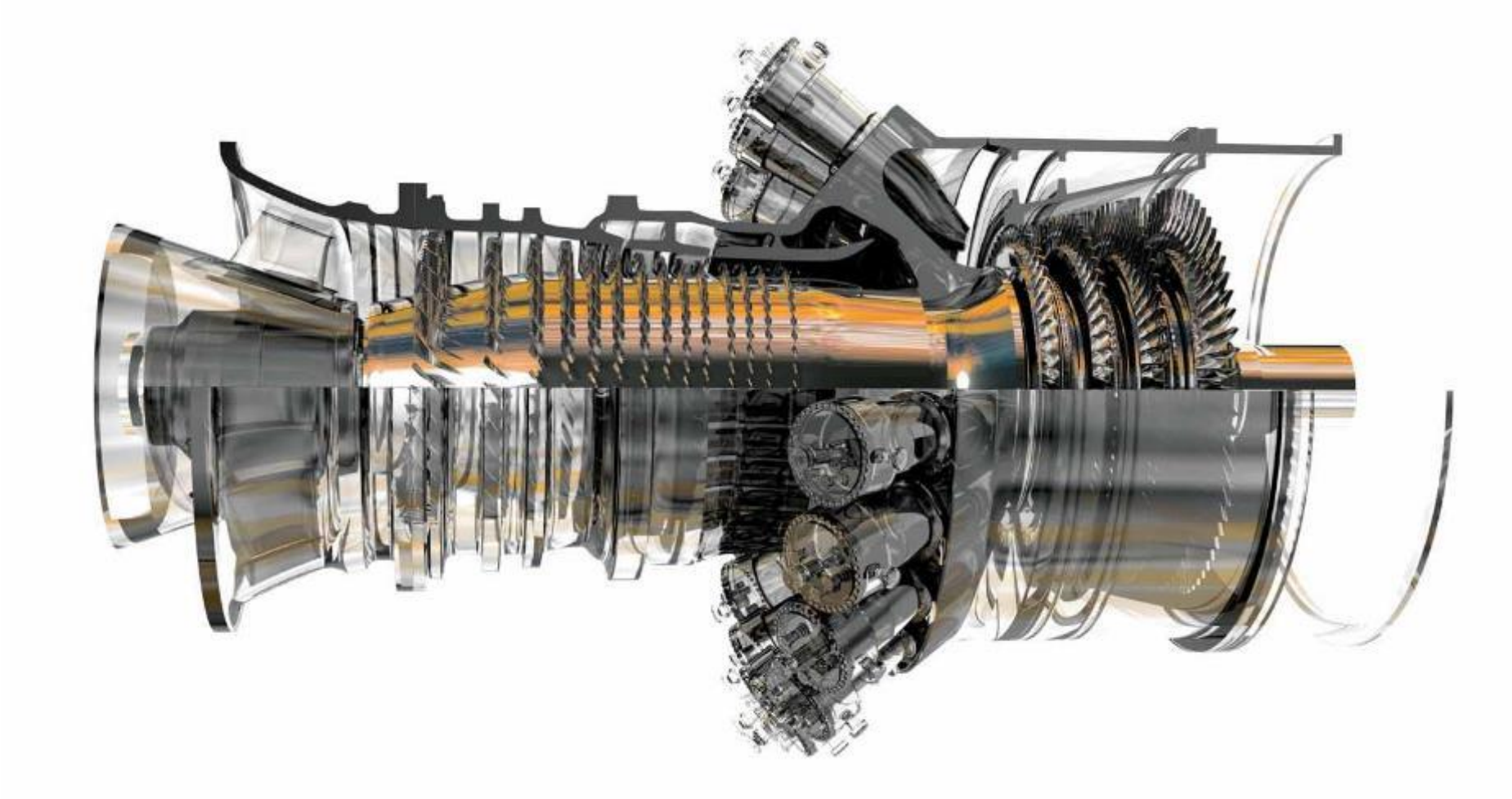
Advanced Skills

07th January 2025

Ennova Technologies

ennova-cfd.com

General Electric 9HA Combustor



GE 9HA gas turbine combustor simulation geometry

Overview

- Models are extremely large so makes interactive training difficult as many tasks can takes many minutes to complete
- Talk about the overall process for preparation for Charles
- Many years of Empirical development as to what has been successful
- Overview of the Strategy
- Develop Ennova skills necessary
- Charles Combustor models are large and the surface mesh ends up around 70m elements alone. We want to make sure the final mesh is consistent geometrically (adequately resolved and no intersections) and has necessary qualities for Charles with correct Mesh Boundary Conditions (for example Periodic etc.)

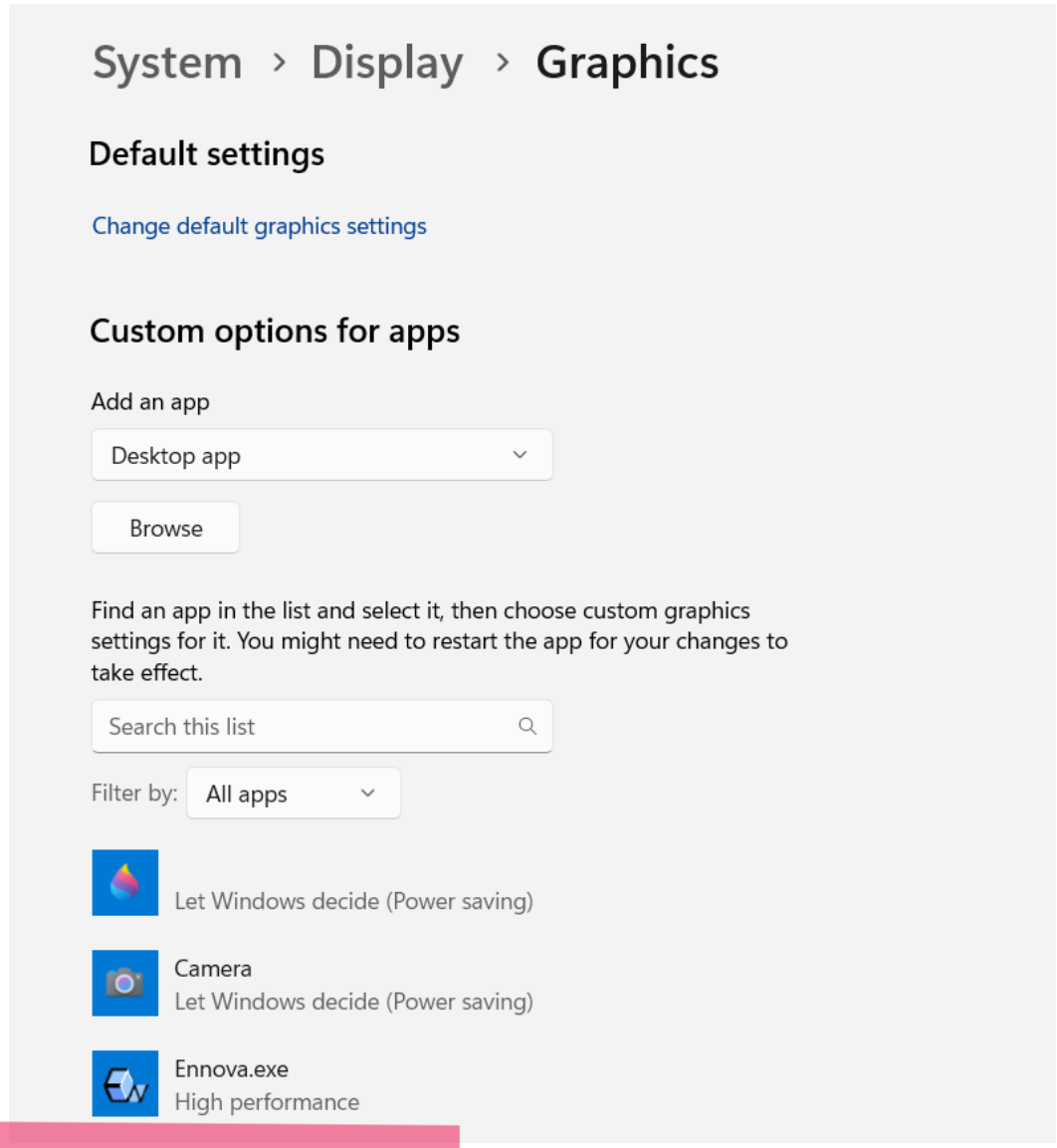
Overall Strategy

- CAD preparation and UG Model
- Hardware Choice for Ennova – Graphics , OS settings
- License Management
- Before Ennova – The CAD database
- Ennova Architecture
- Repair
- Surface meshing
- Techniques for trouble shooting

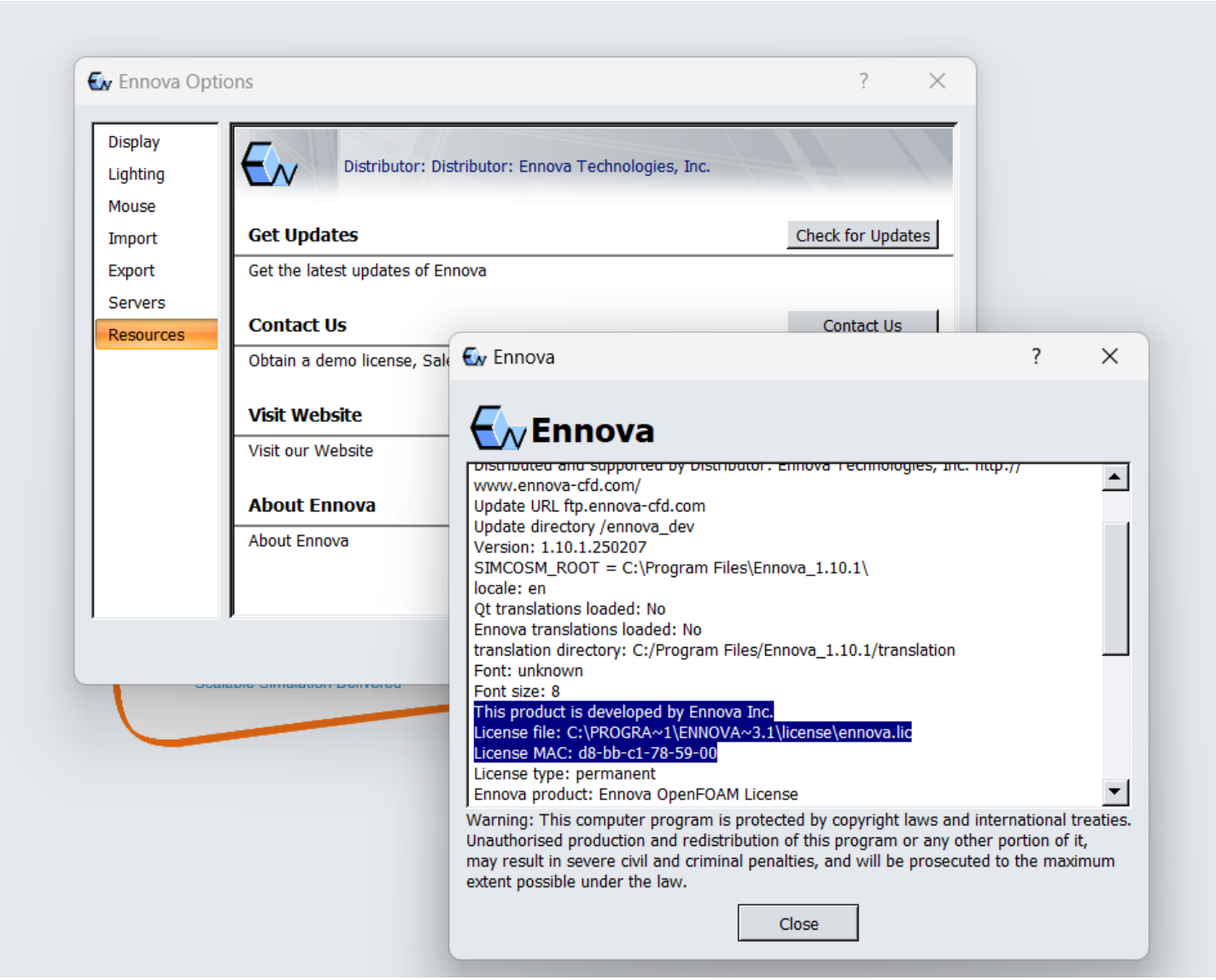
Cad Preparation

- Always try to work with the native UG model if possible
- Want all geometry to be tagged for the CFD Analysis Boundary Conditions
- Different ways to store the tagging , we have done our best to recognize these names based on history of GE models. CAD software is not static so if you encounter problems here we are able to re-train Ennova based on any new CAD changes

Graphics Card Setup – **Do not let Windows Decide**

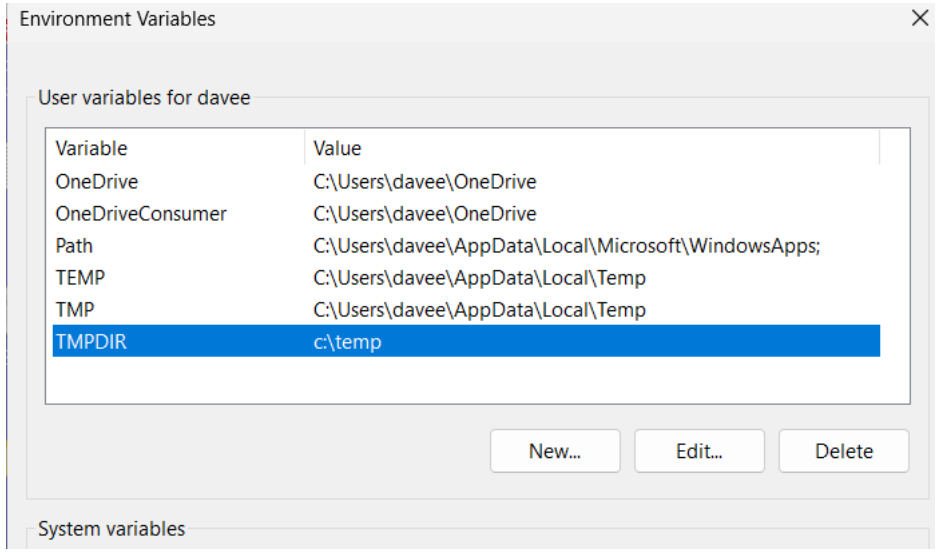


License Management & Software Install



Working Space For Ennova

- Environment Variable TMPDIR



Selection Help

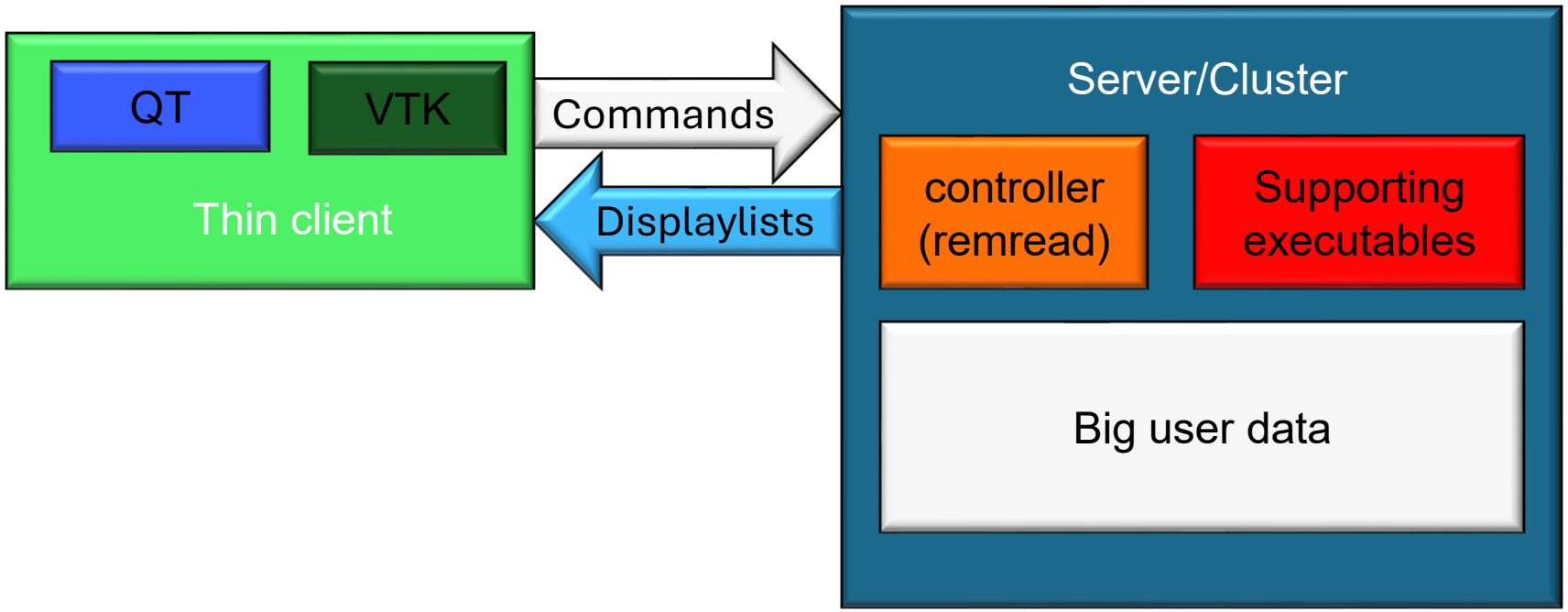
- The Combustor models are large making selection and viewing problematic
- We have developed many methods to help the user
  - Hot Keys , f for face , e for edge and n for node
  - Edge display can be restricted by face or group

Running Ennova Commands from the Command Line

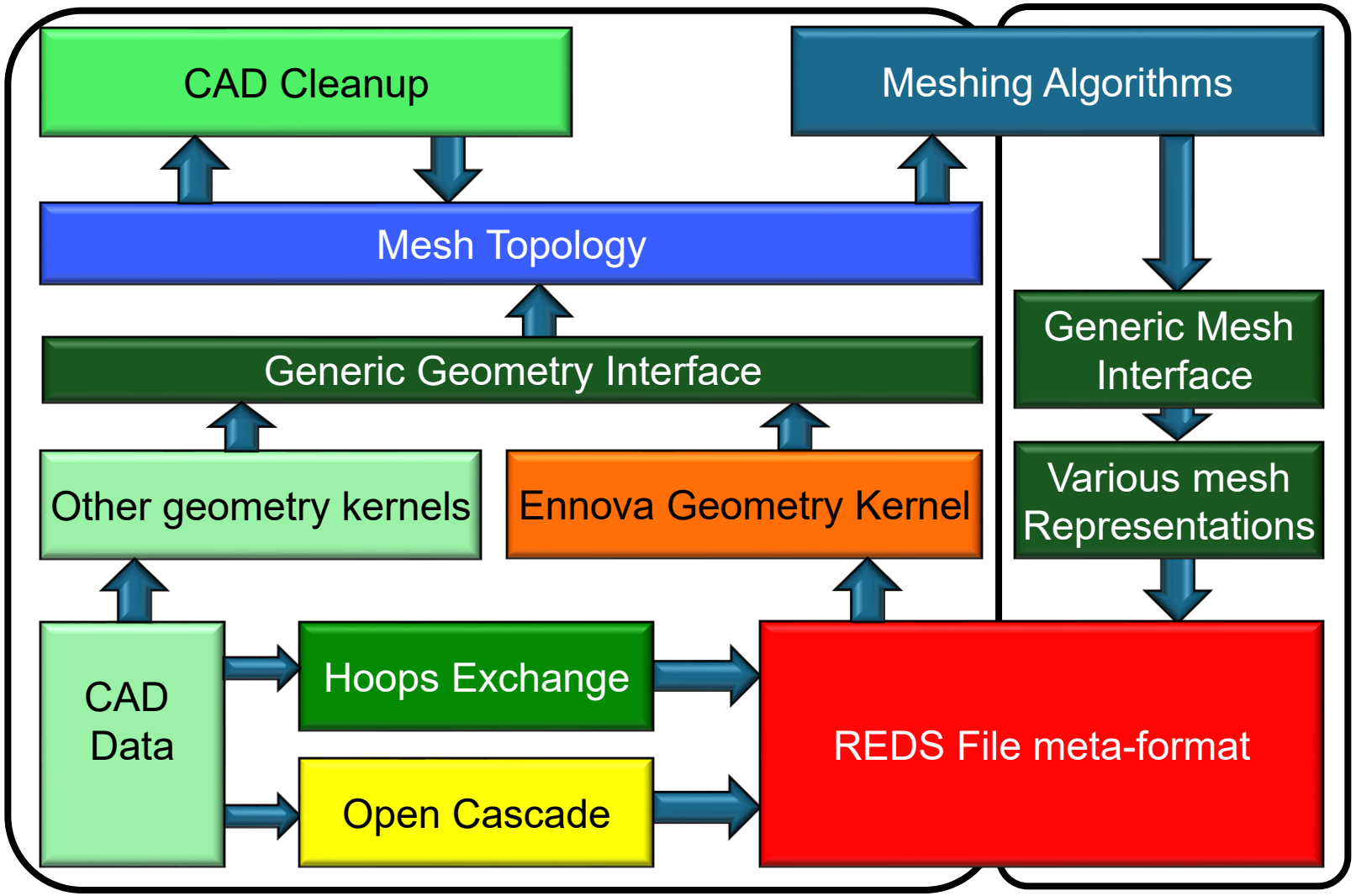
- Example : writeFluent.exe

```
C:\Program Files\Ennova_1.10.1\bin>writeFluent.exe  -help
Usage: writeFluent [inputfile [ outputfile ] ] [options]
Options:
  -i <inputfile>:   the input file
  -o <outputdirectory>:   the output directory
  -s <scale factor>:   scale the mesh
  -v <volume name>:    extract a volume
  -d <decimal places>: output float precision
  -ascii:   export in ascii format
  -space:   replace spaces in group names by underscores
  -auto:    extract volumes automatically
  -explode: export each volume to a separate file
  -of: output file
  -notime: do not print timing
Example:
  writeFluent -s 0.001 -d 8 -v fluid -v solid -i mesh.reds -o mesh.msh
  Description:
    -s 0.001: converts millimeters to meters
    -d 8: write numbers with 8 decimal places in ascii output
    -v fluid: "fluid" is a volume domain
    -v solid: "solid" is a volume domain
    -i mesh.reds: input mesh file; an ascii "*.hum" is acceptable too
    -o mesh.msh: Fluent mesh file
    -ascii: write output in ascii format
```

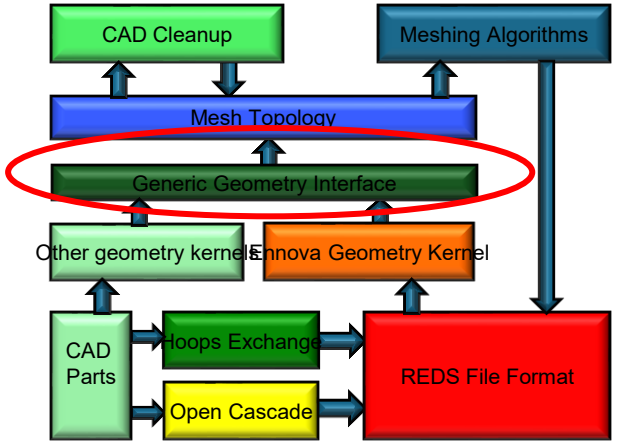
ennova's client/server architecture



server architectural components

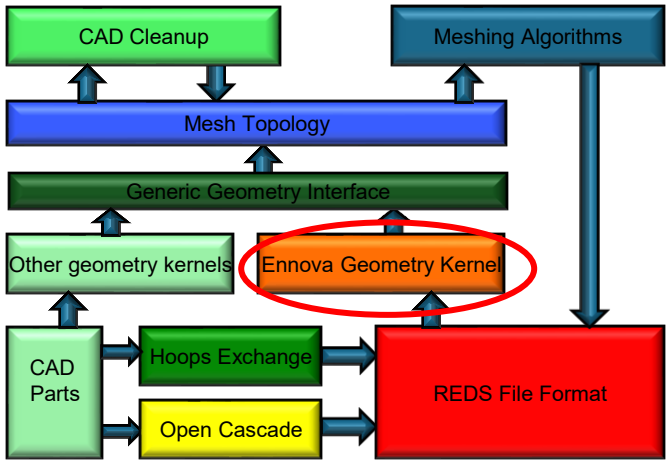


generic geometry interface



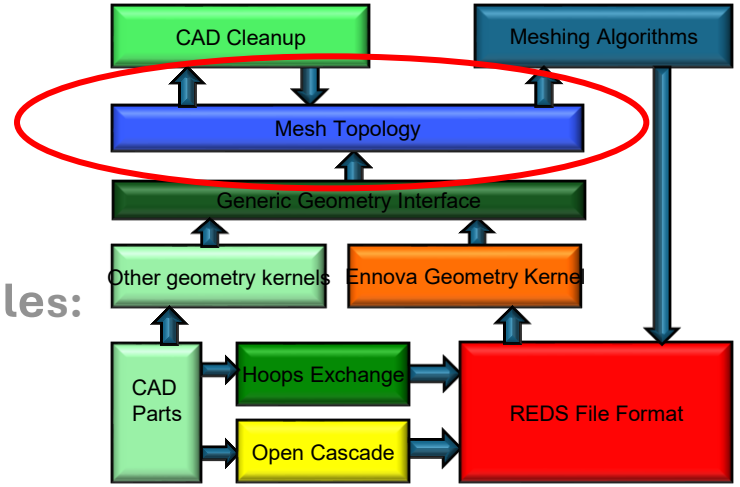
- need to have the ability to port code to different geometry kernels
- also supports access to geometry kernels not running in same address space or even same computer
- supports parallel access
- supports CAD assemblies

ennova geometry kernel



- holds B-splines, STL data and analytic surfaces/curves
- fully thread safe
- high performance search/project/intersect.
- optimized for queries as opposed to creation.
- all queries on B-Splines/analytics to unit in the last place (ULP)-- no tolerances!

ennova mesh topology

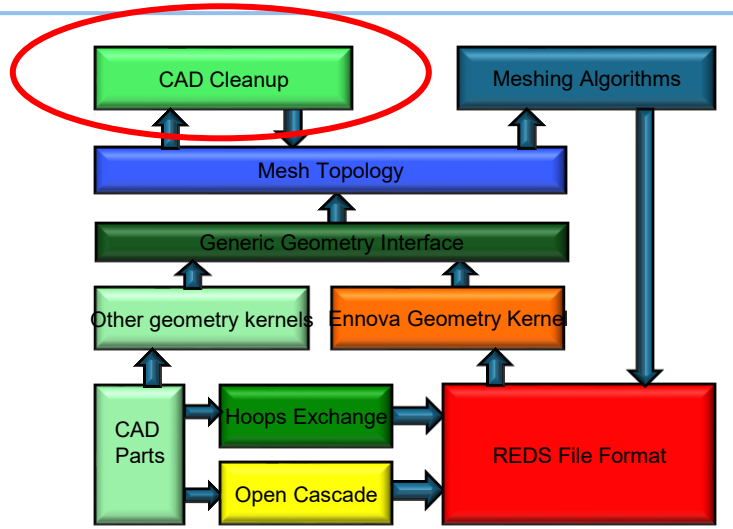


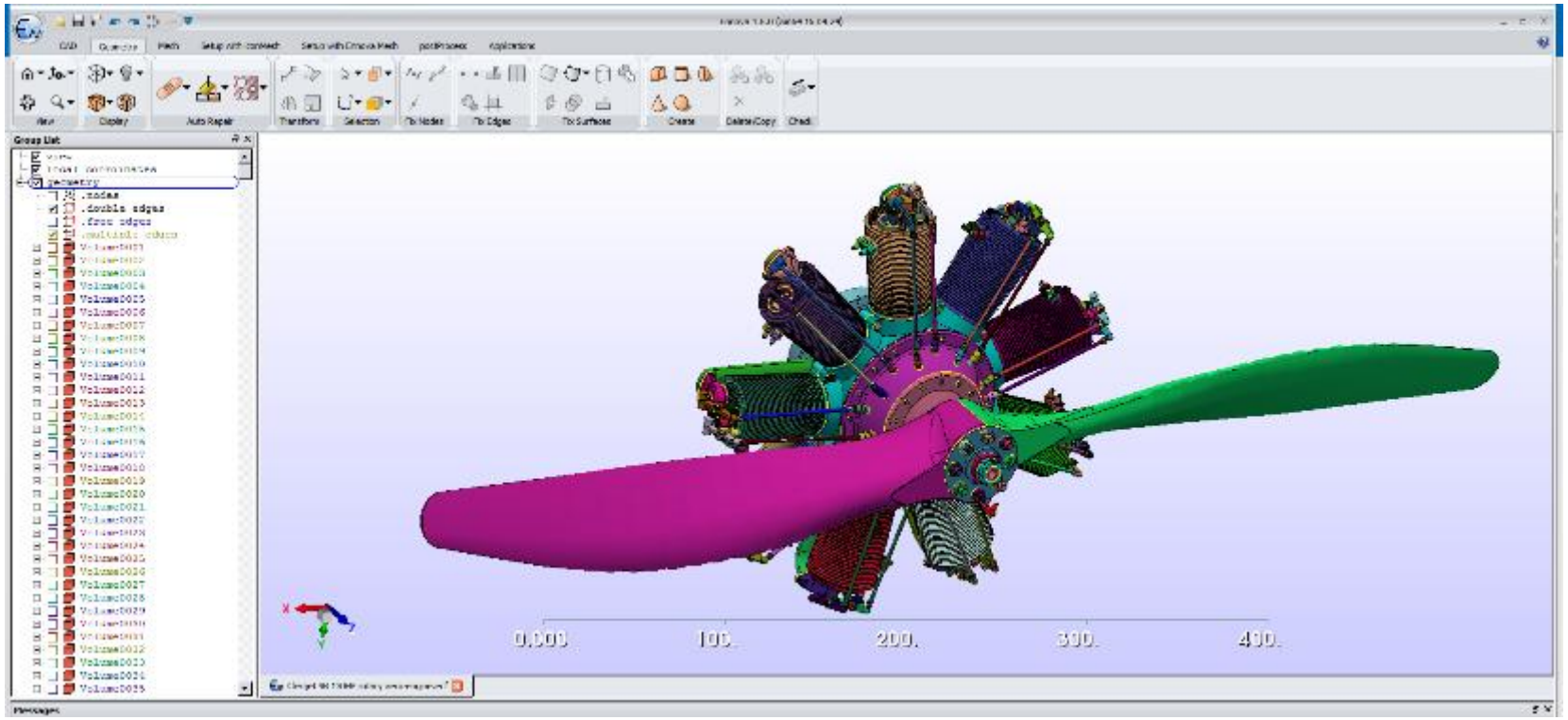
a data layer between CAD and mesh that enables:

- merging/splitting of edges/faces
- interval assignment
- structured faces
- swept volumes
- structured volumes
- booleans between parts from different CAD systems
- has no coordinate data
- maintains the integrity of the original CAD
- enables parallel meshing
- analogous to ICEM HEXA Blocking

CAD cleanup

- based on ennova mesh topology
- fully automatic
- performs independently of user tolerance
- targeted for mesh creation





ennova topology mesher

- follows geometry cleanup
- decomposes space into structured and unstructured faces and structured, swept and unstructured volumes
- naturally builds hybrid anisotropic meshes
- runs in parallel

ability to “look under the hood”

- Most capabilties are available from the unix/windows command line
- Executables read/write geometry/mesh .reds files.
- Geometry import: CAD file to part.reds
- Mesh output: mesh.reds to solver input file
- prism mesher; tet mesher; volume smoother; dualizer
- stand-alone topology mesher.
- stand-alone mesh viewer/checker.
- reds encoder/decoder.

ability to “look under the hood”

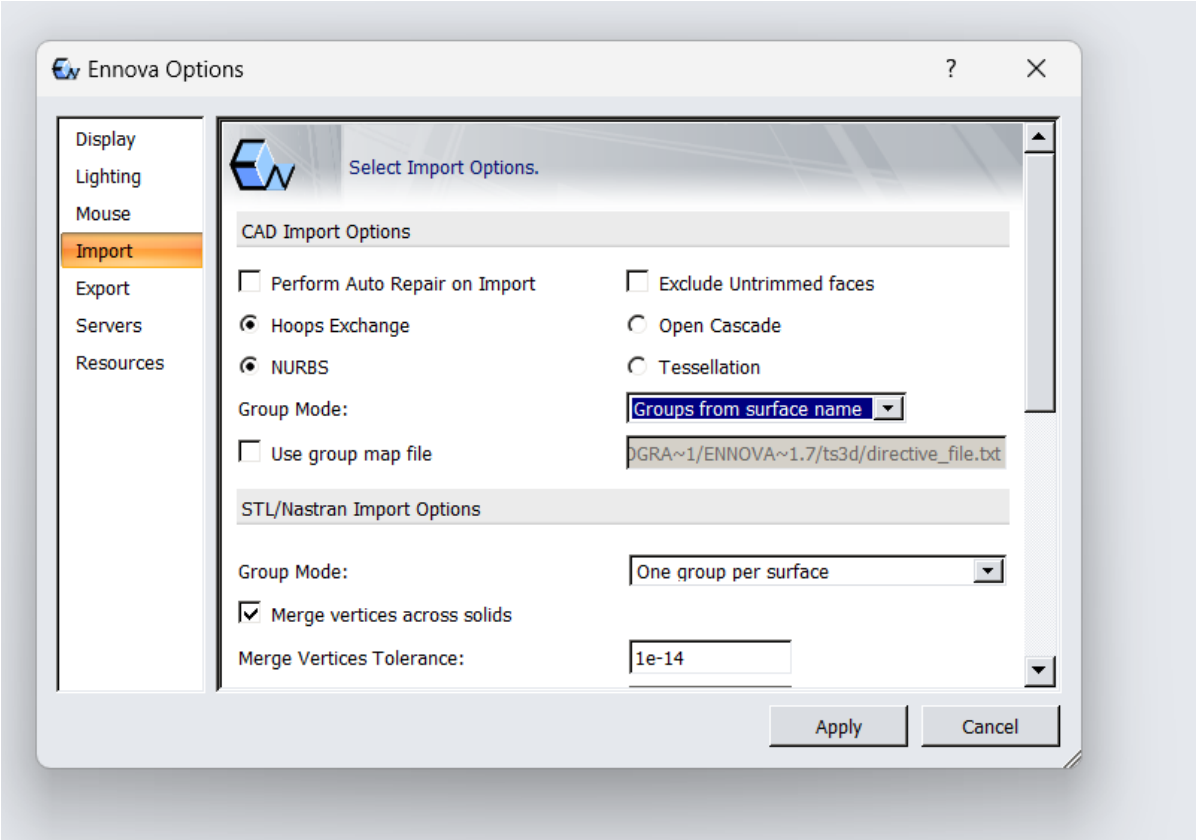
- .ecf (ennova case file) is basically a tar.gz of all of the working files of a session:
  - mesh.reds (current mesh)
  - <partname>.reds (the originally imported CAD part(s))
  - part.db (mySQLdatabse of meshing controls)
    - You can edit this with any mySQL editor.
  - topo.reds (file representation of topological view of CAD)
- Case file implode/explode

Ennova Steps – Putting it All together

- CAD Import

Ennova Import Options

- CAD Data Options
- For GE Combustors from NX Use This Option



Historic NX Model Parameters & Mesh Sizing For Combustors

- Over the past few years we have standardized on the following parameters for Repair and Surface Mesh for the Combustors.
  - Model in meters
  - Overall Geometry Repair Tolerance = 1e-6
  - Maximum Mesh Size = 0.05
  - Minimum Mesh Size = 0.05/(2^5) = 0.05/32 = 0.00016
  - Curvature Refinement = 12

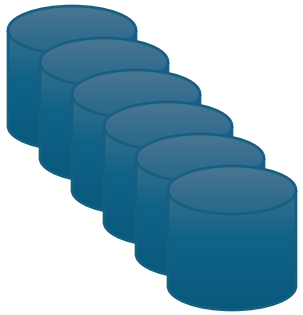
Understanding the Repair Functionality

- Merge Tolerance – We have been using 1e-6 on the combustors
- Stitch Only , merges surfaces and does not do face intersection
  - If the trimmed surfaces are good (Usual) ,this is the preferred merge
- Stitch and Intersect - For bad quality geometry (Not usual in this case)
- Find Volume - Only use once the model is believed to be correct. For large models this is a very expensive command.

A final Word about Ui vs Batch Mode

- The Gui is a generally designed to handle these complex models
- However under these circumstances you will find batch / command line useful
  - Sometimes more options are available by command line or easier accessible (For example writing the final export mesh to disc is more convienently done from script)
  - Scripts can be native to Ennova or Python may be used
  - Sometimes during debugging problematic geometry a command line is more verbal with messages . -v and –V (For verbose) can be used
  - Scripts can be re-used at a later date.

ennova’s approach in Practice



Raw CAD Data, step, iges ,stl, proprietary cad formats

→

Topology Based CAD Repair

→

Best Practices Meshing

→

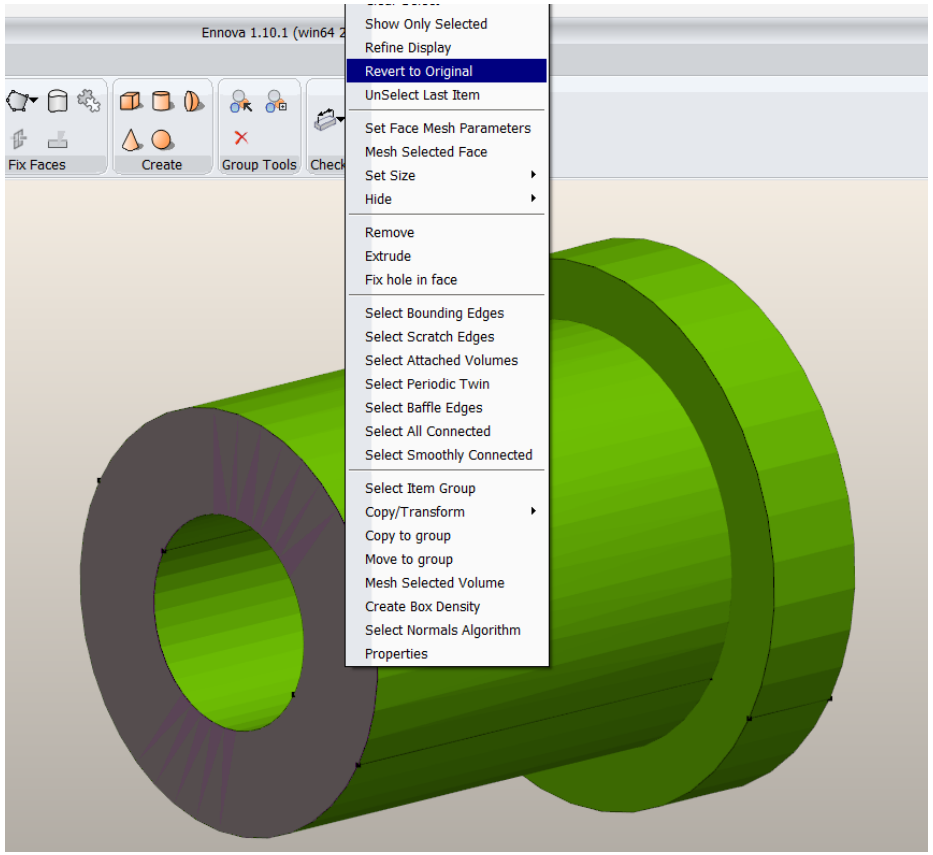
Industry Standard Formats & Direct Solver Support:

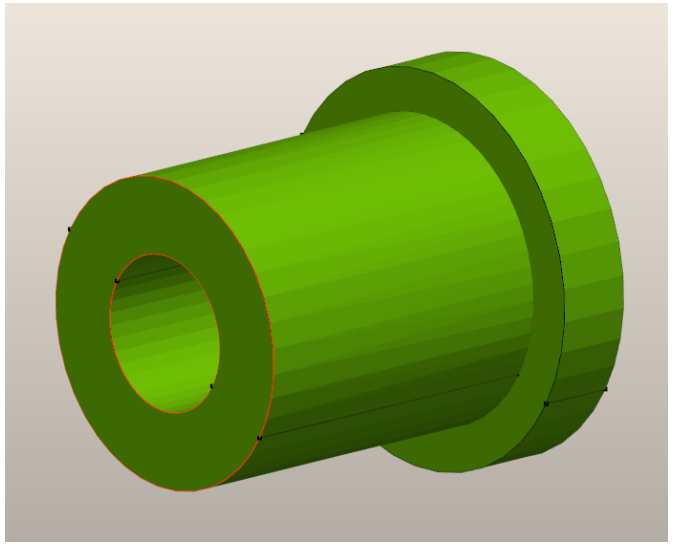
CGNS,UGRID,MSH,STARCCM, NASTRAN,CFD++, UNS, etc.

Example – Repair for Charles

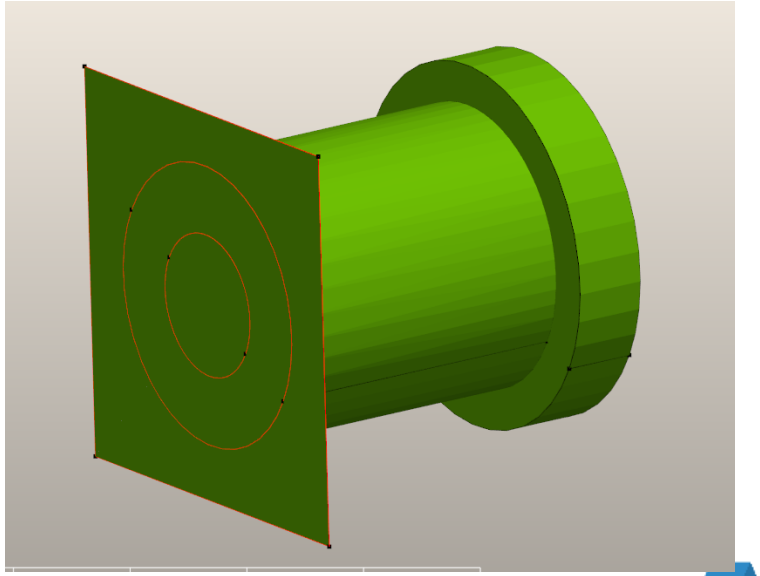
- Load Model , groups
- Stitching
- Untrimming
- Splitting
- Volume Detection
- Periodic Boundaries

Untrimming / Revert to Original (Catia WaterBottle_Nozzle)



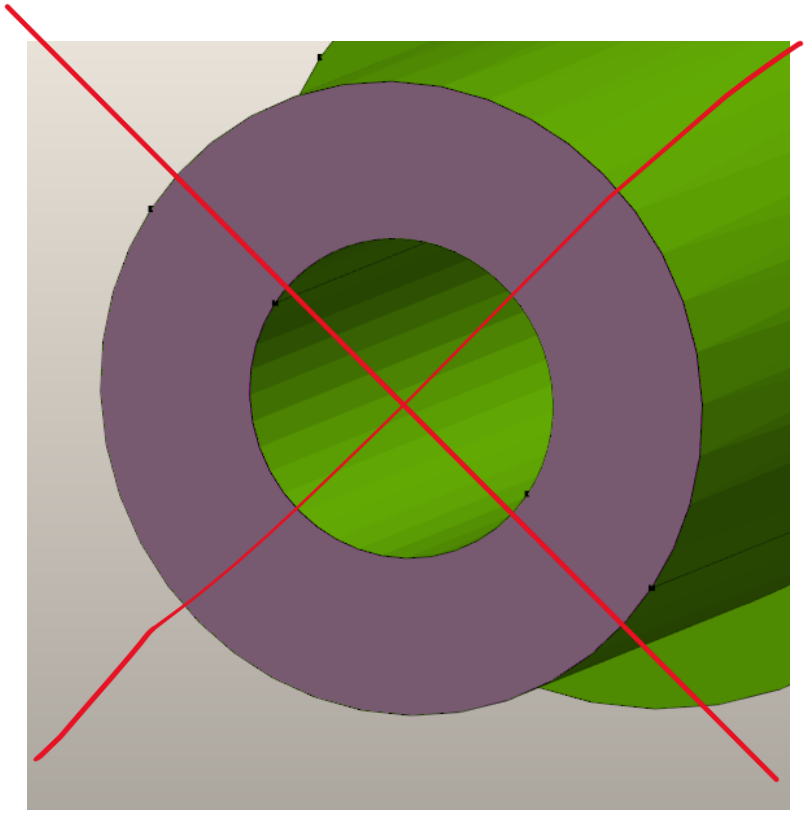


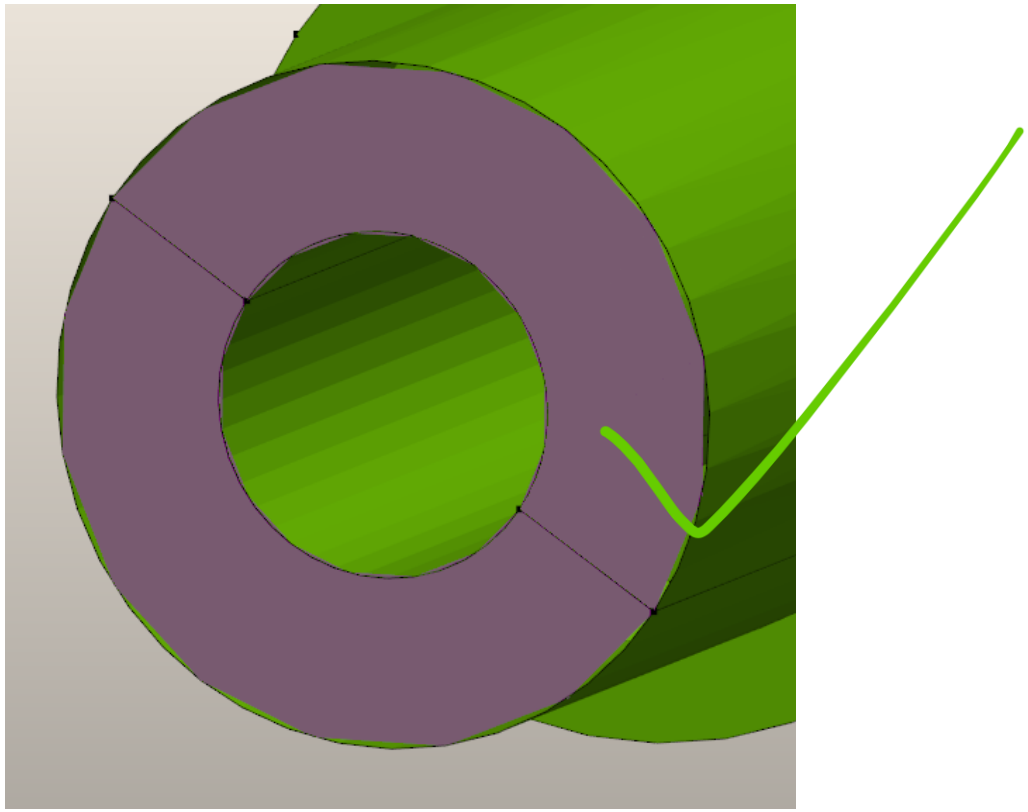
Repeat Command

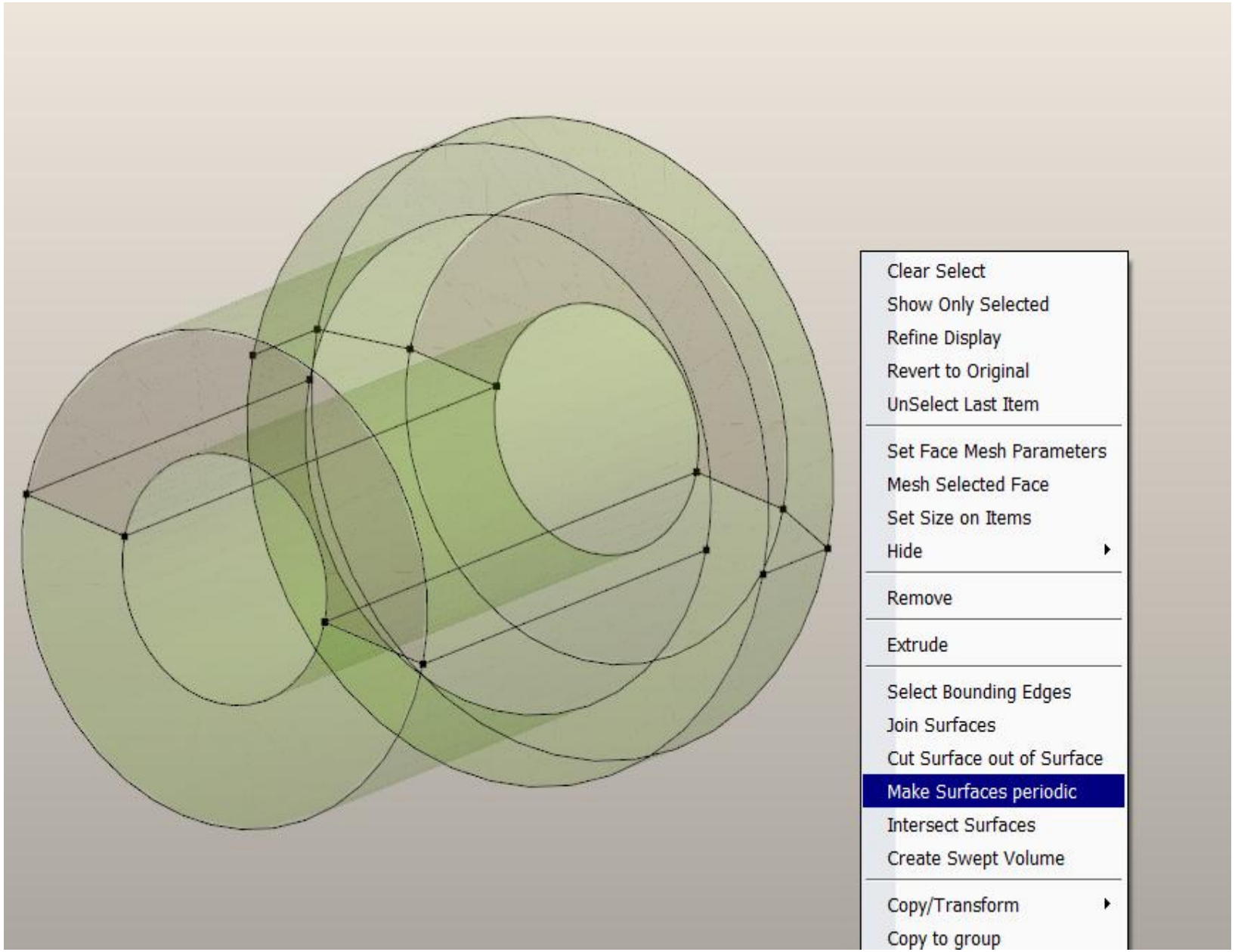


Periodic Boundaries

- All Periodic Faces Must be split into simply connected Faces






Clear Select
Show Only Selected
Refine Display
Revert to Original
UnSelect Last Item
Set Face Mesh Parameters
Mesh Selected Face
Set Size on Items
Hide
Remove
Extrude
Select Bounding Edges
Join Surfaces
Cut Surface out of Surface
Make Surfaces periodic
Intersect Surfaces
Create Swept Volume
Copy/Transform
Copy to group