Release Notes
Grid*Pro v6.5*

Snapshot

PURPOSE: Capture images of workspace or entire GUI.

How to use :

1. Rotate screen to the desired position.
2. Go to File menu.
3. Select Entire UI or Graphic window from the drop down
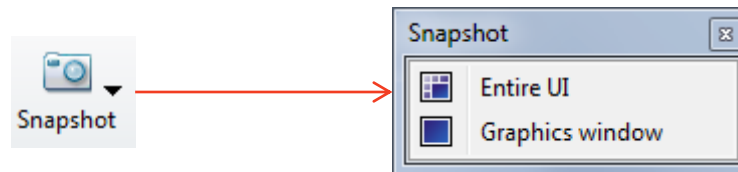4. Save the captured image in JPEG, PNG or BMP format.

Figure 1: Snapshot button drop down list

Set property

Easily assign property to the surface using right click menu.

How to use :

1. Right click on the desired surface whose property has to be changed.
2. Select 'Set property' from the drop down.
3. Select the desired property in the dialog box and click 'ok'

Note: The selected property will be assigned to the surface and it will be written to the conn file while running Ggrid.
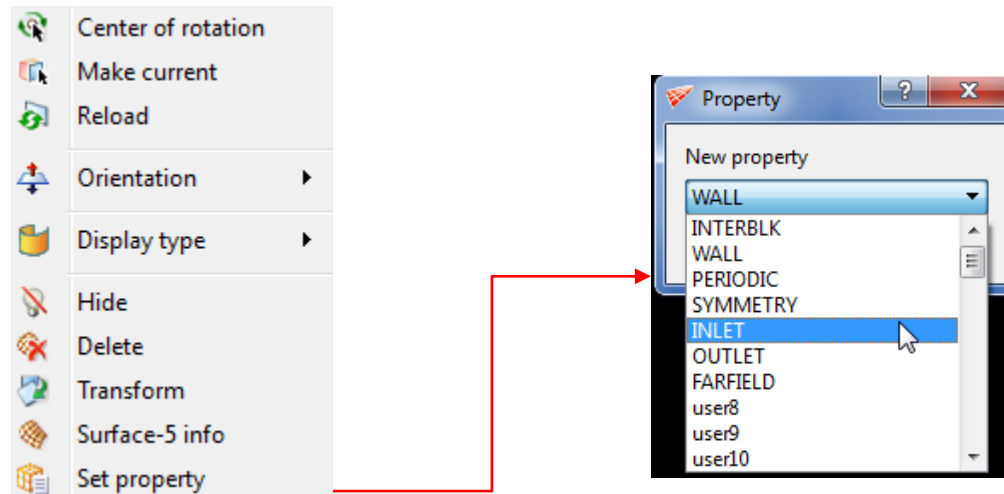

Figure 2: Set property button in Surface right click menu

Scaling

Input the get workplane parameters automatically using 'Get workplane center' button.

How to use :

1. Position workplane to the desired location.
2. Open Scale dialog box and click on 'Get workplane center'.

It will automatically find the center of the workplane and enter it in the Center field of the dialog box.
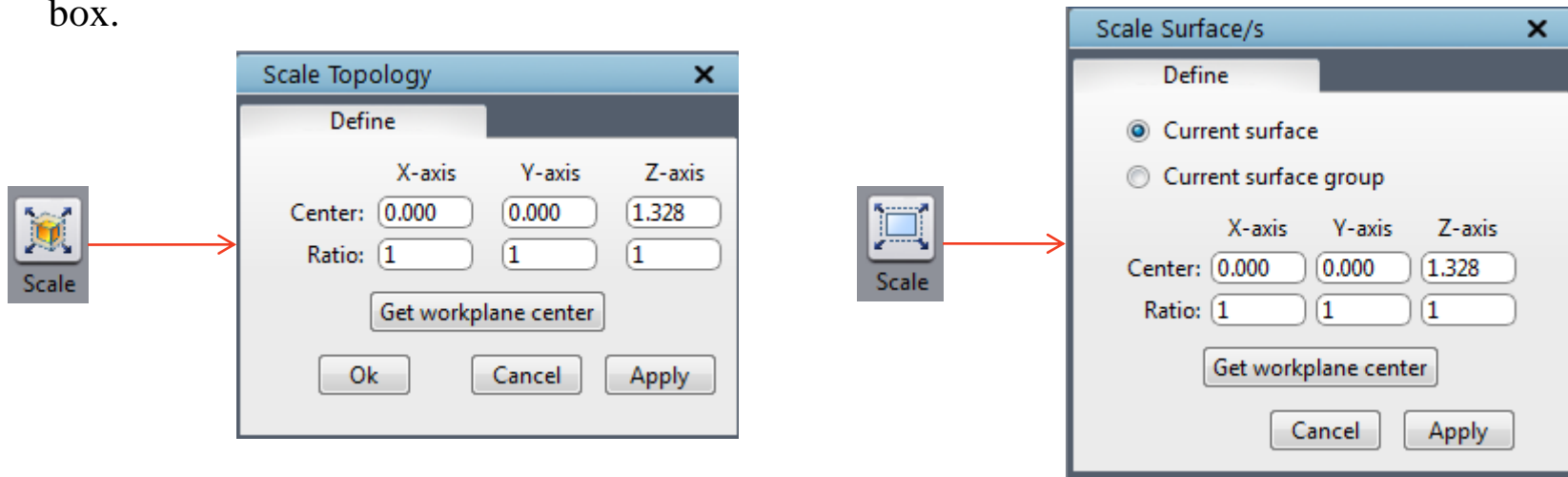

Figure 3: Scale dialog box of Topology and Surface tab

Holes

PURPOSE: Automatically create topology and internal surface for the pipes connecting two holes.

How to use :

1. Click on Holes button.
2. Select the appropriate surface id's to which the pipe surfaces are connected.
3. Select the corner group to which the outer layer of resultant topology should be grouped.
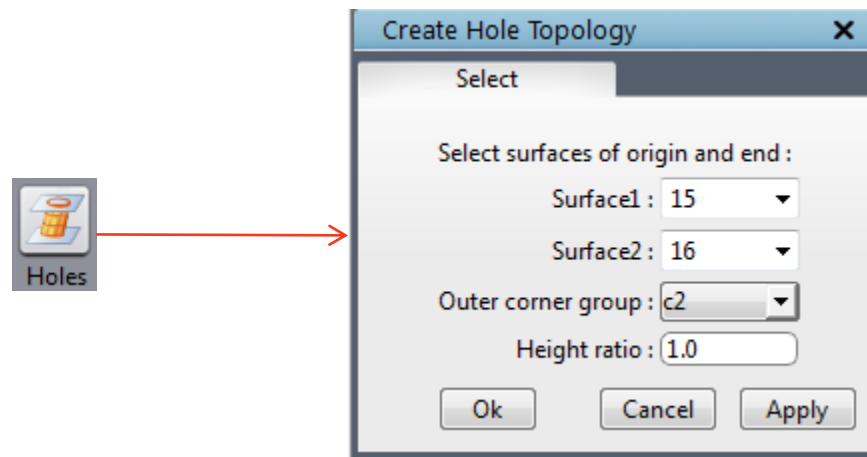4. Enter the Height ratio (internal surface height) and click 'Apply'.
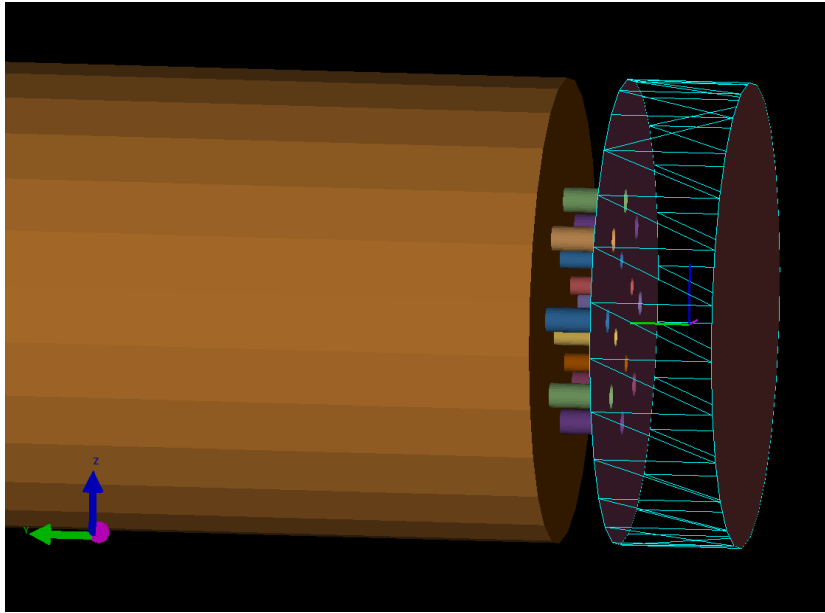

Figure 4: Holes dialog box
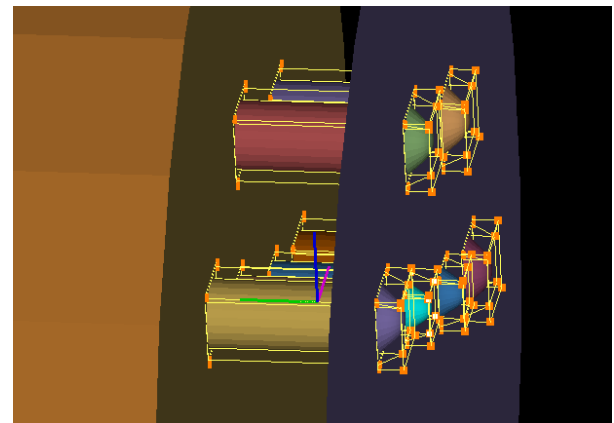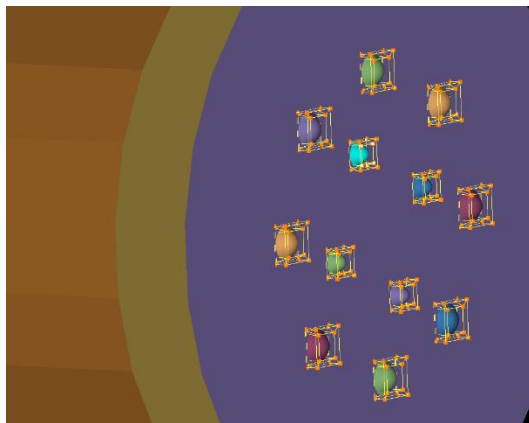
Figure 4a: Input surfaces

Figure 4b: Output

Schedule File

PURPOSE: Save schedule file in different names.

How to use :

1. Go to Ggrid→Schedule.
2. Enter the desired input parameters including the name to which it has to be saved.
3. Click 'Apply'.

Application: When ggrid has to be run from command line with different schedule parameters other than default.
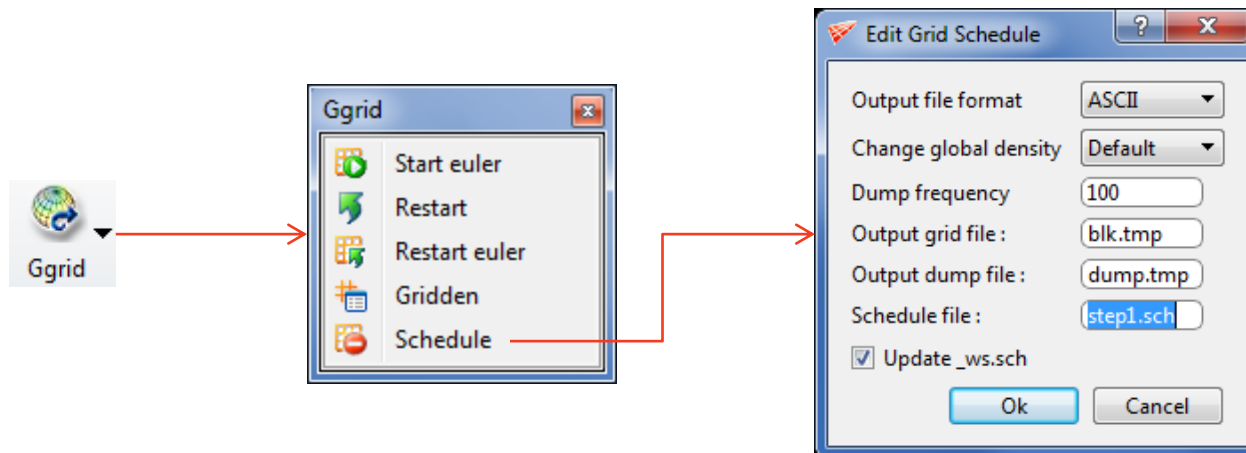

Figure 5: Schedule dialog box

Restart Ggrid

PURPOSE: Stop the existing gridding process and restart a new process.

How to use :

1. Click on Start while Ggrid is running.
2. A dialog box opens with two options, Restart and Cancel.

Selecting 'Restart' will stop the existing process and start a new process from sweep 1.
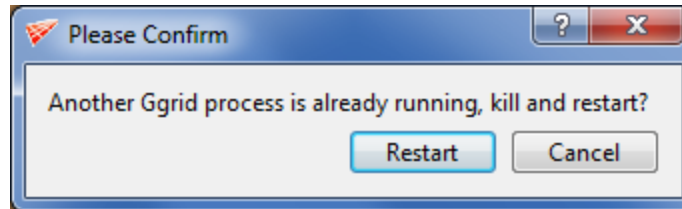

Figure 6: Confirmation dialog box

Save grid sheets

PURPOSE: All active grid sheets can be saved as a single quad file.

How to use :

1. Switch ON the grid sheets to be saved .
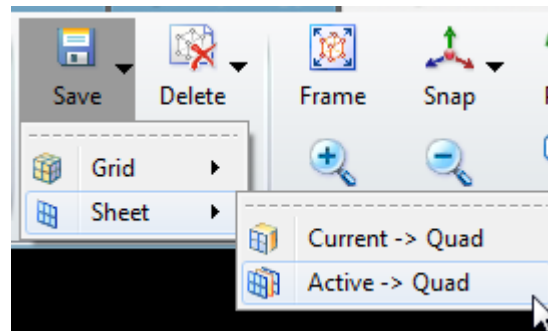2. Click on *Save →Sheet →Active->Quad*

Figure 7: Save active grid sheets

Faces

PURPOSE: Create grid sheets using Surface id's.

How to use :

1. Identify the surface id whose grid sheets you want to create.
2. Click on 'Faces' and switch to 'Specify' tab.
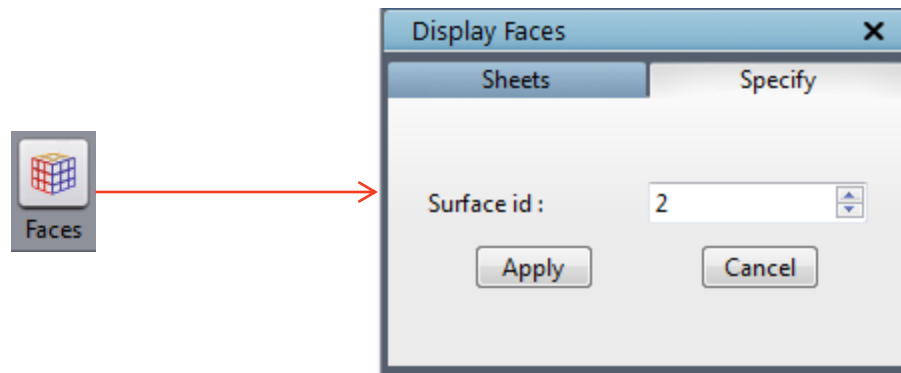3. Enter the surface id as n+1, where 'n' is the surface id displayed in Surface/Topology tab.

Figure 8: Specify Tab

Labels

PURPOSE: To generate tagging and property for the surface sheets using the surface ids.

How to use :
1. Load the grid.
2. Click on 'Labels' button and 'Backup and continue' in the confirmation dialog box.
3. Check 'Properties' section in the Property tab to view the labels.

Every surface sheet will be tagged using its surface id and default properties(Wall and Interblk) will be applied to each of them.
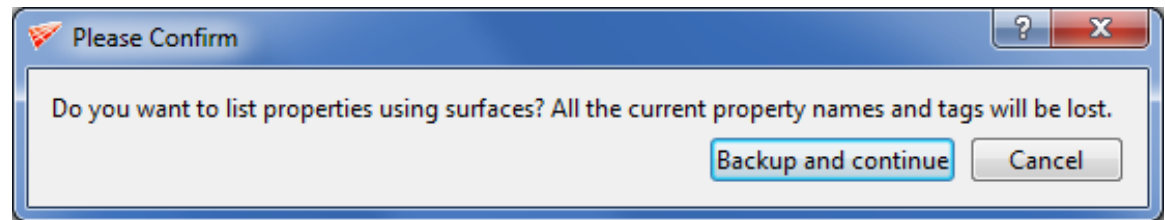

Figure 9: Labels button

Create property sheet

PURPOSE: To create new property sheet with labels

How to use :

1. Load the grid.
2. Switch to *Property* tab and Go to ***Create*** tab in *Solver* section.
3. Select the Property to be created from the solver section and enter the label name.
4. Click 'Apply' to create the selected property.
5. Click on '.' or '+' and select the block face sheets to add them to the selected property.
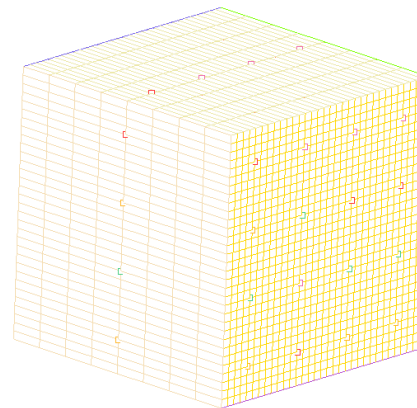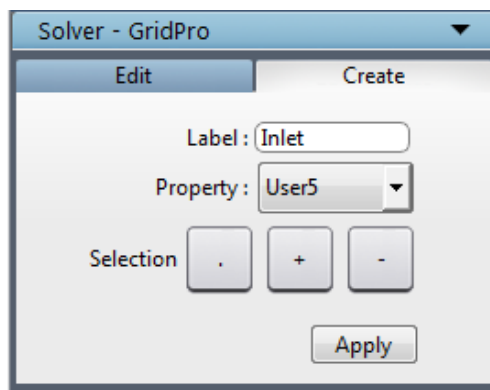

Figure 10: Labels dialog box and block face centers of a grid

Add labels to property sheet

PURPOSE: Edit the Property labels in the Properties section.

How to use :

1. Click on any property displayed in the Properties section.
2. Modify the labels in the edit tab in Solver section.
3. Click 'Update' button to save the modified labels.

Note: The first name represents the boundary condition of the grid sheet and the second name represents its property label.
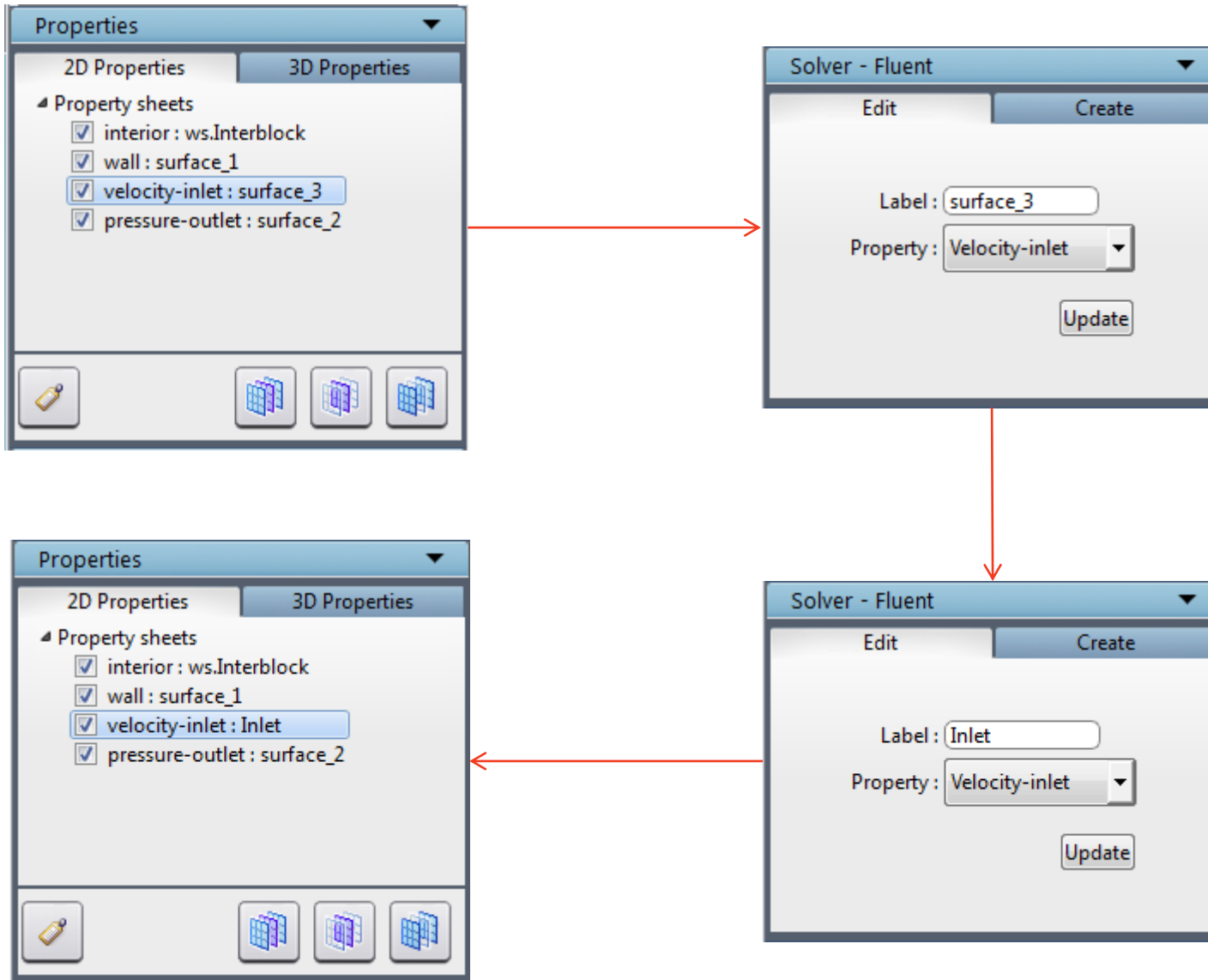
Figure 11: Modifying the property labels

CFD++ solver

PURPOSE: Export Grid*Pro* grid to CFD++ solver format

How to use :

1. Select CFD++ from File→Export→Grid→CFD++

Note: It will create all the necessary files inside 'CFDPP_Files' directory

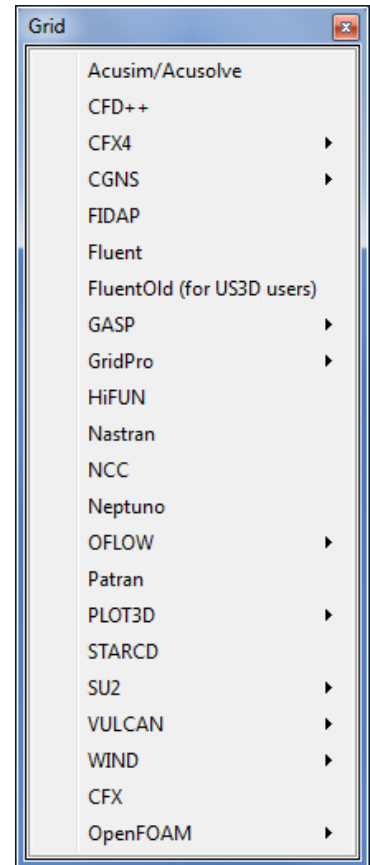*Similarly you can export other solvers, For e.g., Neptuno, OFLOW, etc.

Figure 12: Export → Grid drop down list

CFX solver

PURPOSE: Export Grid*Pro* grid to CFX solver format.

How to use :

1. Once you have the Grid*Pro* grid, open CFX pre-processor UI.

2. Go to Import→Mesh

3. Select 'User import' option from the File type.

4. Navigate to GridPro installation directory and select 'ws' executable in the "Exec Location" input field

5. Enter these arguments "change_format –ifn" in the "Exec Arguments" field.

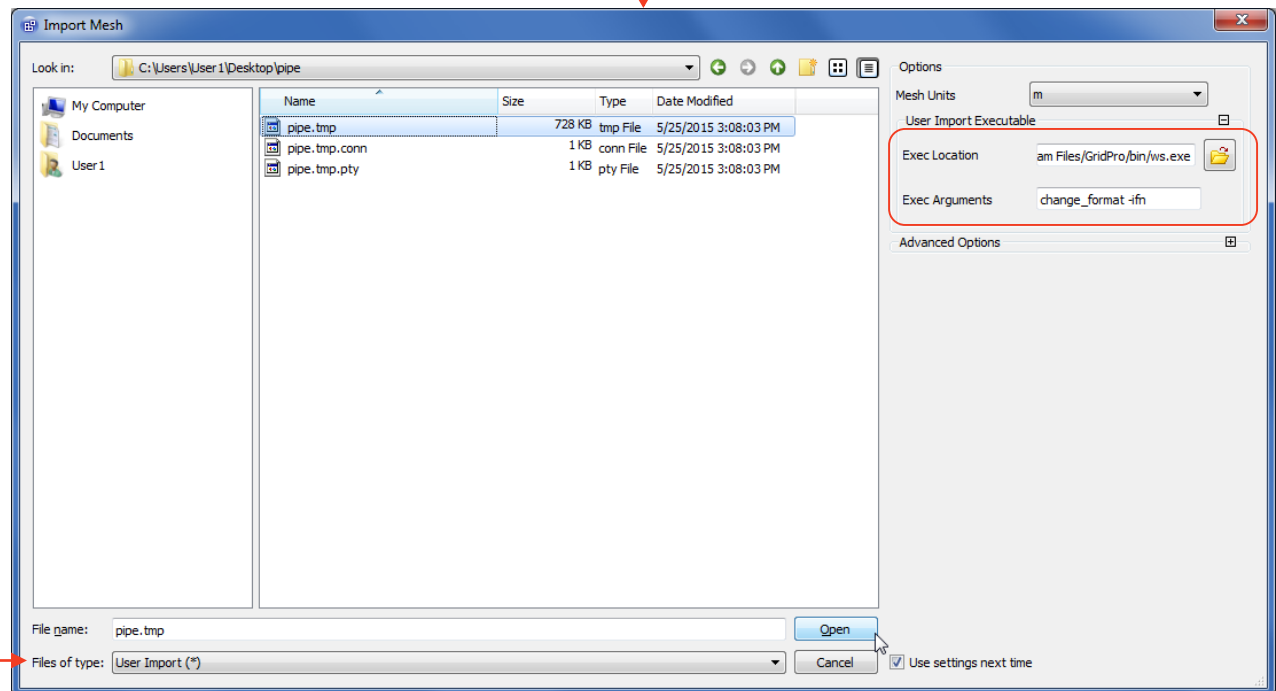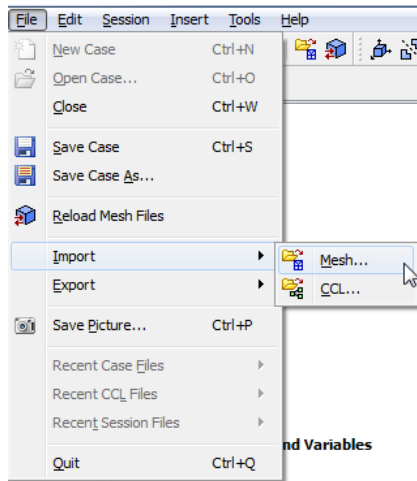6. Then select the GridPro grid to be imported into the CFX.

Figure 13: Import CFX

Import hex grid

PURPOSE: Import a Hex grid into Grid*Pro* UI(WS)

How to use :

1. Select Hex from File→Import→Grid→Hex

2. Select the input file name with extension '.hex'

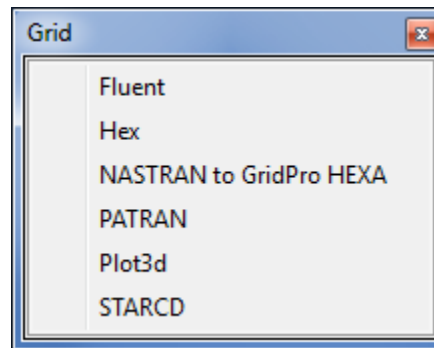Note: The Hex grid will be converted to Grid*Pro* grid format and load it into the UI.

Figure 14: Import → Grid drop down list

C-Edge

PURPOSE: View the connectivity of the geometry.

How to use :

1. Load the geometry either in IGES or STEP format.

2. Click on 'C-Edge' button and select the connectivity that you would like to view. By default all the edges are selected

Note: Connectivity defines the number of times an edge has been used in the geometry. A connectivity of 2 represent that the geometry is well connected.
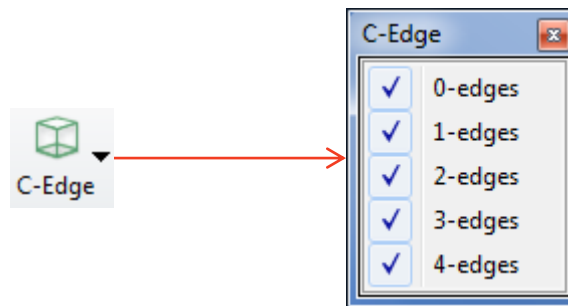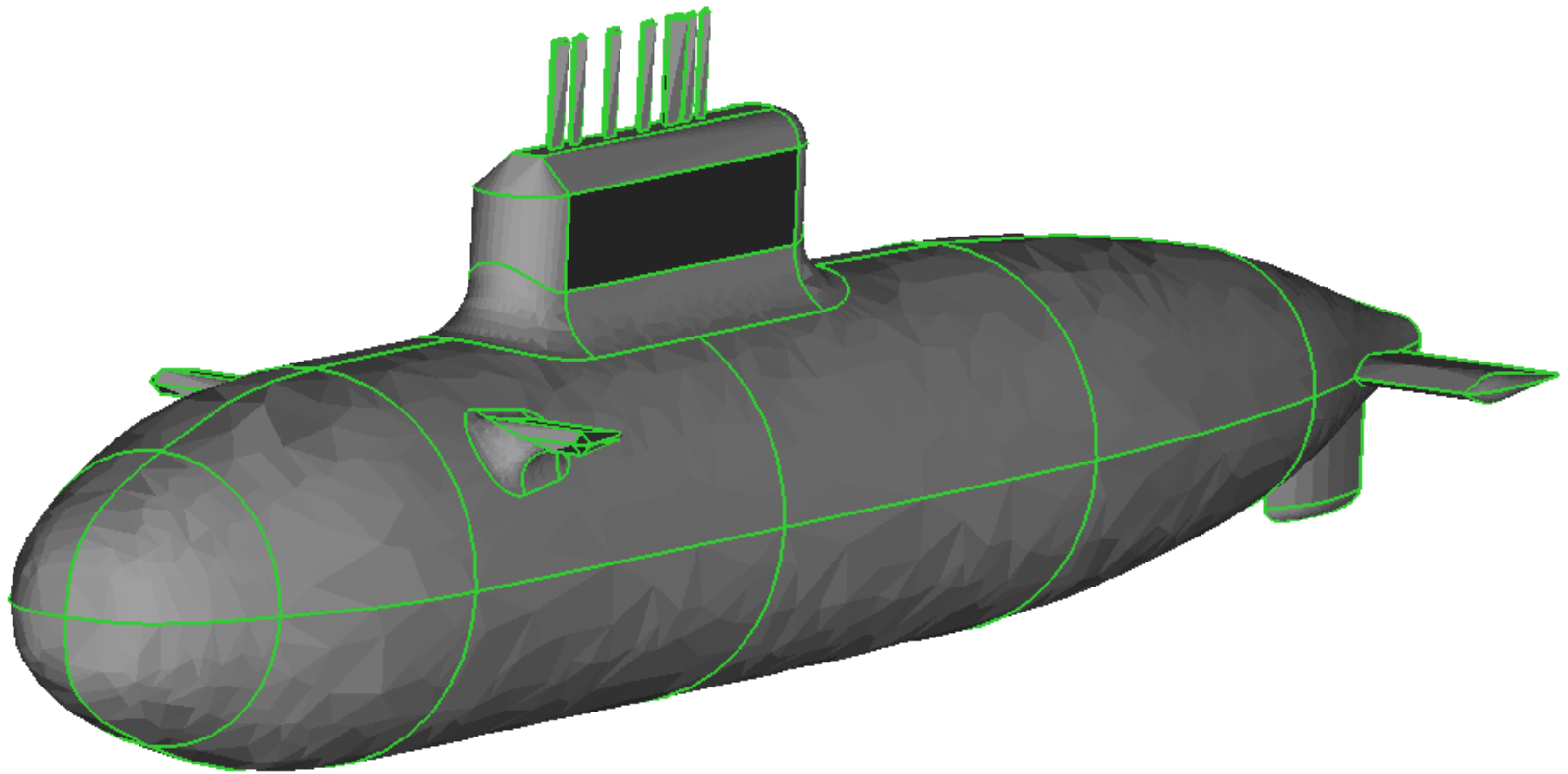

Figure 15: C-Edge drop down list

Figure 15a: Geometry connectivity lines

Toggle Geometry Elements

PURPOSE: Toggle geometry related objects

How to use :

1. Go to Toggle Display section.

2. Click on Geometry toggle tree.

3. Check/uncheck the checkbox of the desired element to toggle them.
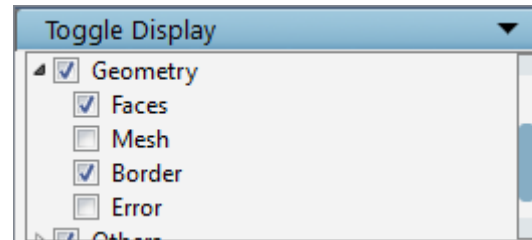

Figure 16: Toggle Display section

Geometry Faces

PURPOSE: Show/Hide the faces of the geometry.

How to use :

1. Load the geometry either in IGES or STEP format.

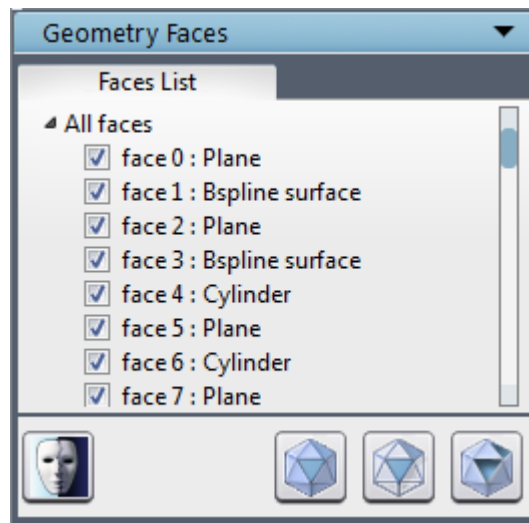2. Go to 'Geometry Faces' section to view different faces of the geometry.
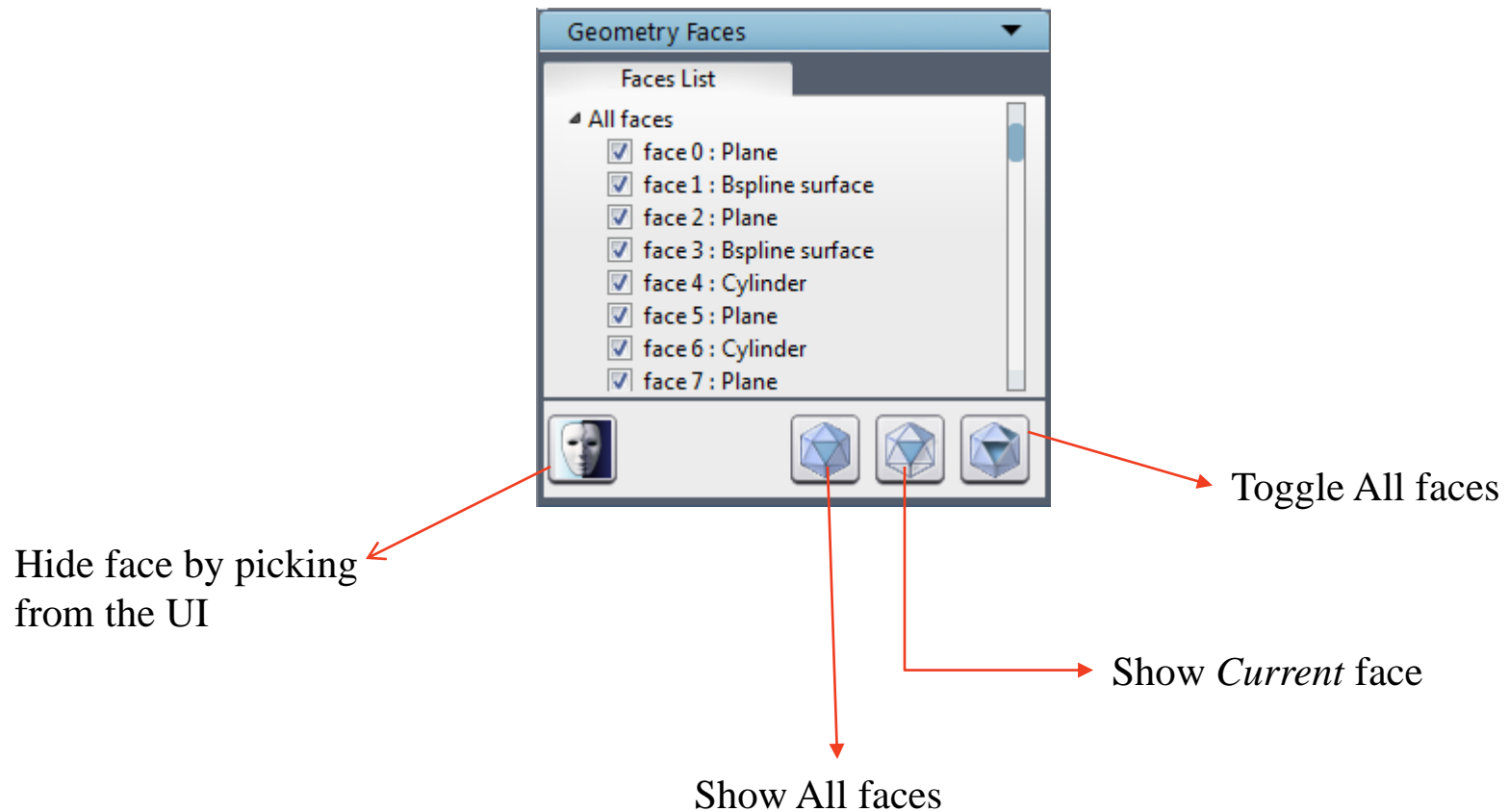

Figure 17: Geometry Faces section

Geometry Faces

Faces List

⊿ All faces
- ☑ face 0 : Plane
- ☑ face 1 : Bspline surface
- ☑ face 2 : Plane
- ☑ face 3 : Bspline surface
- ☑ face 4 : Cylinder
- ☑ face 5 : Plane
- ☑ face 6 : Cylinder
- ☑ face 7 : Plane

Hide face by picking from the UI

Toggle All faces

Show *Current* face

Show All faces

Figure 17a: Toggle buttons in Geometry Faces section

Heal

PURPOSE: Repair the geometry to fix small edges, fix spot strip faces and sew faces using a tolerance value.

How to use :

1. Load the geometry in IGES or STEP format.
2. Click on 'Heal' button.
3. Select necessary options to fix and click 'Apply'.

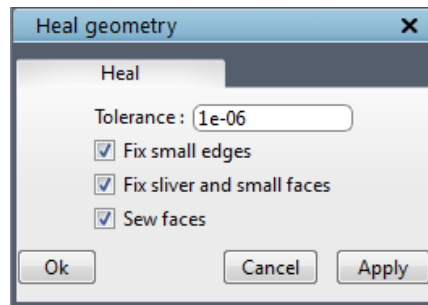Note: Modify the tolerance value if there are any faces/edges not fixed.

Figure 18: Heal Dialog box

Mesh

PURPOSE: Create surface mesh for the imported geometry.

How to use :

1. Load the geometry for which the surface mesh has to be generated.
2. Make sure the geometry is repaired.
3. Click on 'Mesh' button and enter the necessary inputs.
4. Click 'Apply' and wait until the surface mesh is displayed. This may even take few minutes depends on the input parameters and size of the geometry.
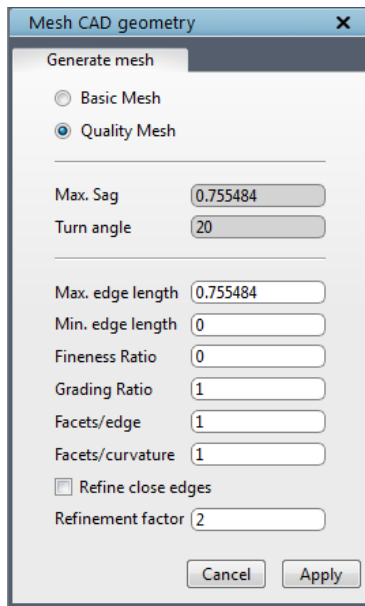
Mesh CAD geometry

Generate mesh

○ Basic Mesh
● Quality Mesh

Max. Sag	0.755484
Turn angle	20

Max. edge length	0.755484
Min. edge length	0
Fineness Ratio	0
Grading Ratio	1
Facets/edge	1
Facets/curvature	1
☐ Refine close edges	
Refinement factor	2

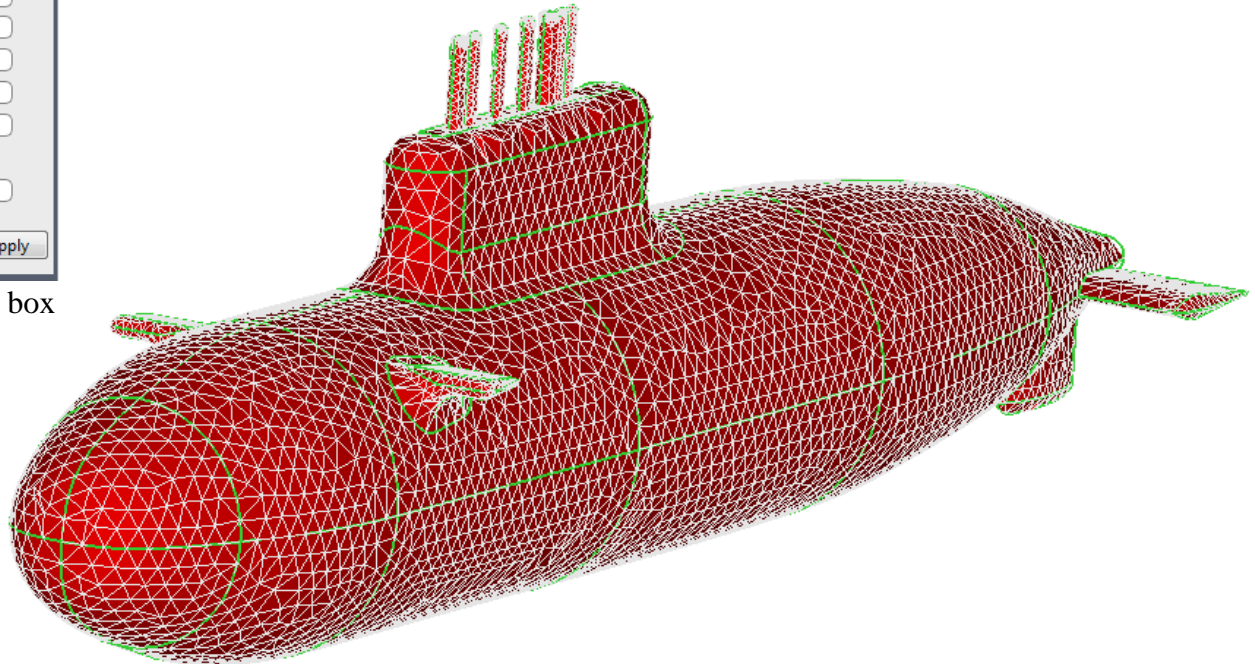Cancel   Apply

Figure 19: Mesh dialog box

Figure 19a: After creating surface mesh for the geometry

Local Mesh

PURPOSE: Create surface mesh locally for certain faces.

How to use :

1. Right click on the face whose surface mesh has to be changed.
2. Select 'Set face mesh size'
3. Enter the new mesh size and click 'ok'.
4. When the surface mesh is created using 'Mesh' button, this new value will be considered for that face.
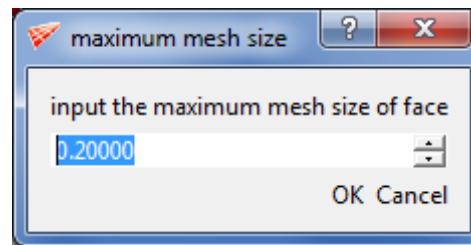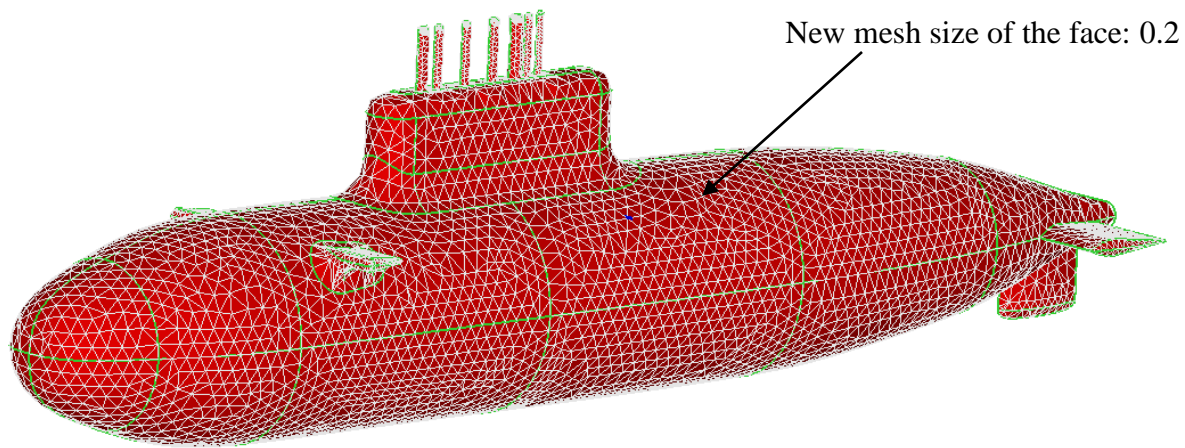

Figure 20: Mesh size dialog box

New mesh size of the face: 0.2

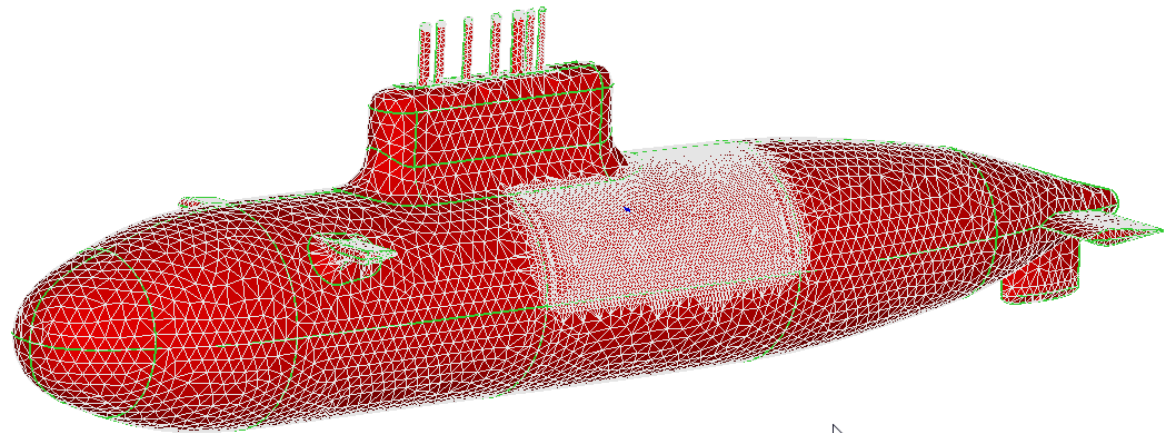Figure 20a: With default mesh size

Figure 20b: With new face mesh size

Tolerance

PURPOSE: Set tolerance value to view the edges to be merged in the geometry.

How to use :

1. Go to View menu and select 'Tolerance' option.

2. Enter the desired tolerance value and click 'Ok'

Based on the given tolerance value, possible red coloured lines are merged together and display it in green colour. This is a preview of running 'Heal' option with the same tolerance.
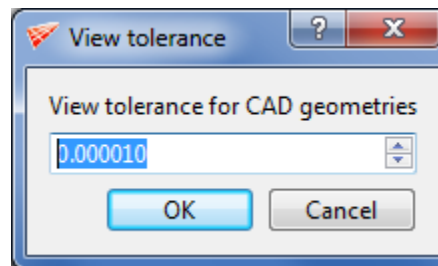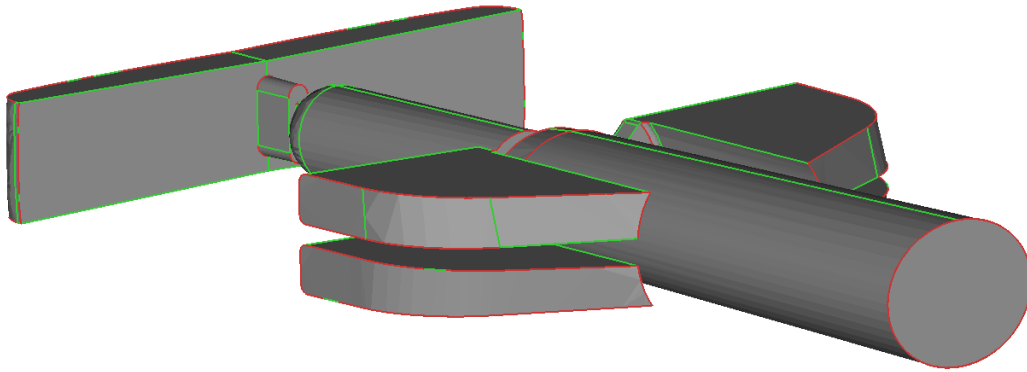

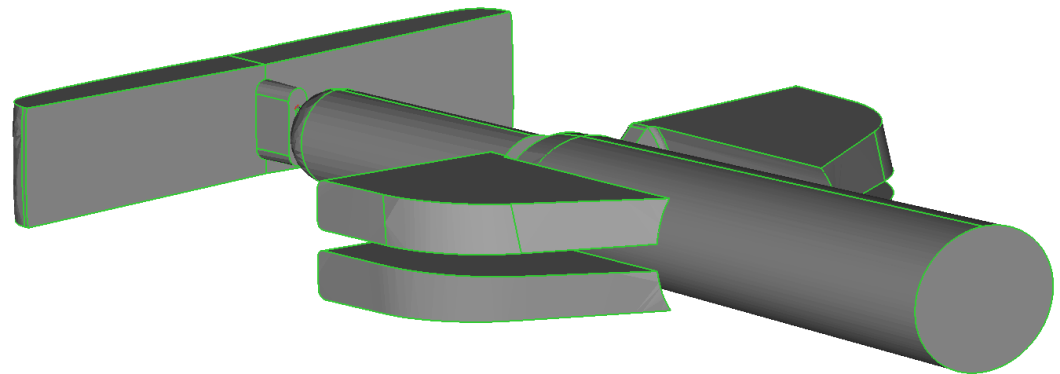Figure 21: Tolerance dialog box

Figure 21a: Tolerance 1e-06

Figure 21b: Tolerance 1e-01

For any Queries
please contact
customerassistance@*gridpro.com*