cādence®

Quadcopter Tutorial

Rapid Adoption Kit (RAK)

Product Version: Pointwise 18.5 R1
January 2022

Copyright Statement

Contents

Purpose

This purpose of this RAK is to guide you through the basic process of creating a surface grid and volume grid using the available automated tools: *Automatic Surface Mesh* and *Automatic Volume Mesh*. This RAK is solely intended to show the basic usage of these tools and is not designed to provide a thorough description of all the available options and parameters.

Audience

This document is intended for Pointwise users who wish to use the automated tools available to effortlessly create surface and volume grids.

Terms

CFD   Computational Fluid Dynamics

CAE   Computer-aided Engineering

Overview

This tutorial contains a brief overview of the operations necessary to set up an unstructured surface mesh and viscous volume mesh about a quadcopter geometry using our automated tools: *Automatic Surface Mesh* and *Automatic Volume Mesh*. This exercise will help you to become familiar with Pointwise's flexible and powerful approach to automated grid generation.

Geometry

The geometry to be meshed is one half of a simplified quadcopter. Since this tutorial will focus on the steps necessary to create a high-quality viscous unstructured volume mesh, you will use a geometry that has already been cleaned up by removing gaps and overlaps and assembled into a single watertight database model with appropriate meshing regions (i.e. database quilts).
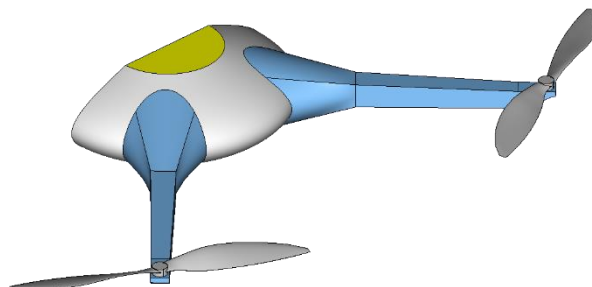

Figure 1: The half quadcopter geometry that will be used in this tutorial.

Topology

The topology will consist of unstructured hybrid surface domains that describe the center-body, the arms, the simplified motors, and the propellers of the quadcopter. Additional hybrid unstructured domains will be created to represent the farfield used to assemble the final voxel block.
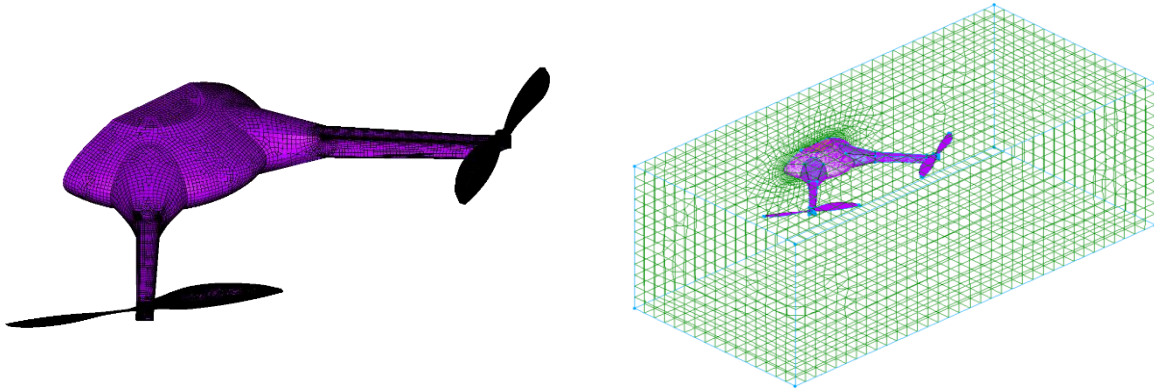

Figure 2: The final topology of the viscous mesh to be constructed. The surface grid is shown on the left side and the volume grid is shown on the right side.

Database Import

The initial file used in this tutorial can be downloaded from this page. You may want to save this file to a local directory before using it.
The geometry you will use in this example is a Pointwise project file named *quadcopter-ASM-AVM-Initial.pw*.

1. Select the *File* menu option (upper side of the Pointwise interface).
2. Select *Open* (second from the pull-down list).
3. Select *quadcopter-ASM-AVM-Initial.pw* from the file browser.
4. *Open*

You should now have a representation of the geometry in your *Display* window similar to *Figure 1* on the previous page.

CAE Selection

Before starting a meshing project, it is always a good idea to first set the grid dimension and CAE solver. This helps avoid any situations in which you create a mesh type that is unsupported by your solver.

1. Note in the *Status Bar* that the current CAE solver is *CGNS* and the current dimension is *3D.*
2. *CAE, Select Solver*
3. *OMNIS*
4. *OK*

You should see the name of your currently selected CAE solver, OMNIS, appear in the left-hand corner of the Pointwise status bar.

Surface Mesh Creation

Pointwise offers a wealth of tools to create surface meshes (Review the Pointwise User Manual for details). These are all fantastic tools providing ever more control, convenience, and robust meshing. However, using al these tools independently to create a wonderful set of surface grids can take some time and experience with the software and our recommended best-practices for different cases.
The *Automatic Surface Mesh* tool offers a target driven approach to surface grid generation that encapsulates all our recommended best practices for any case you might be working on. Additionally, the starting point to use this automated tool is a watertight database model with well-defined meshing regions (i.e. database quilts).

1. Select the Database model named quadcopter.
2. Select the *Create* menu option (upper side of the Pointwise interface).
3. Select *Automatic Surface Mesh* (first option in the pull-down list).
   For a detailed description of all the options available in this functionality, please review this section of the Pointwise User Manual.
4. Make sure the *Global* tab is currently selected.
   This panel presents the geometric characteristics of the selected database models. It also contains tools to define the meshing resolution goals, its quality constraints, and some additional advanced parameters to control the characteristics of the resulting surface mesh.
5. In the *Goals* frame, enter *20.0* for *Max. Aspect Ratio.*
   The Max. Aspect Ratio parameter limits the maximum aspect ratio allowed for the anisotropic cells created by the T-Rex algorithm along the 2D T-Rex boundaries. In this particular case, we increase this parameter from its default value of 10.0 to reduce the number of grid points along the spanwise direction of the propeller blades.
6. Select the *Surfaces* tab.
   This panel contains tools for adjusting the display style within the Automatic Surface Mesh command and for specifying Resolution and Mapping filters, which allow you to assign specific target edge lengths for each quilt and specify which quilts receive a mapped meshing treatment (i.e. the resultant unstructured domains are mapped to and retain the appearance of underlying structured domains).
7. In the *Mapping* frame (i.e. lower table), click *New.*
   In this particular case, we will use the tools in this frame to create a new mapping filter to force the number of grid points across the blades' trailing edges to a user-specified specified value.
8. Double click *filter-1* in the *Name* column.
9. Enter *TEs* (for Trailing Edges).
10. In the *Pattern* column enter **TE*.*
    Specifying a pattern will allow you to quickly assign all database quilts with names matching the specified pattern to a new mapping filter. In this particular case, the four trailing edge quilts on both blades are named: Prop-TE-1, Prop-TE-2, Prop-TE-3, and Prop-TE-4; they all match the pattern *TE*.
11. Note that the four trailing edge database quits for the blades have been added to the new filter.
12. Note the following in the new mapping filter:
    a. The database quilts added to the new filter are rendered in a unique color matching the color shown in the *Set* column in the new row.
    b. The number of added database quilts is shown in the # column in the new row.
13. Double click *Automatic* in the *Type* column.

14. Select *Force* from the pull-down list.
15. Enter *5* in the *Subdivisions* column.
16. Select the *Boundaries* tab.
17. Note that the different edge types (i.e. *Curvature*, *Convex*, and *Concave*) are rendered in a unique color in the table and the *Display* window.
18. In the *Convex* row in the table, double click the *Max. Aspect Ratio.*
19. Select the *Off* option from the pull-down list.
    This effectively turns off stretching of the surface cells coming off of the convex edges in the geometry.
20. Select the *Global* tab.
21. Click *Generate Surface Mesh.*

The new surface grid should look similar to the one shown in *Figure 3*.
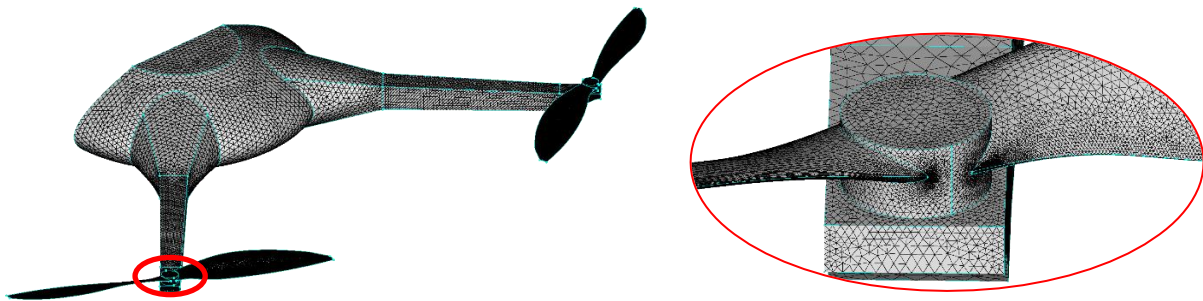

Figure 3: Surface grid created by the Automatic Surface Mesh functionality. On the left side, we show a general overview of the entire surface mesh. On the right side, we show a closeup view of the surface grid around one of the quadcopter's motors.

Before exiting the *Automatic Surface Mesh* functionality, please review the quality metrics presented in the *Global* tab.

22. Make sure the *Global* tab is currently selected.
23. Note the table in the *Display* frame at the very end of the panel.
24. Note that there are surface cells that violate the *Area Ratio* threshold set to 10. Also note that the *Max. Value* is only slightly above the defined threshold.
25. Check on the *Show* option in the *Area Ratio* row.
26. Note that the surface cells violating the defined *Area Ratio* threshold are rendered in red. Also note that these cells are located on the blades' trailing edges (*Figure 4*).
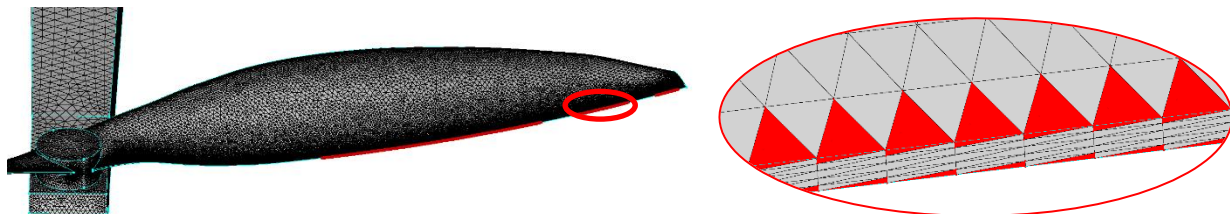

Figure 4: Surface cells violating the defined Area Ratio threshold. On the right side we present a closeup of these cells located in the trailing edge region of the four blades in the geometry.

Even though the created surface mesh is of good quality and could certainly be used for a CFD simulation, in the next section we will review how to resolve the *Area Ratio* problem shown above.

27. Click *OK.*
    This will exit the Automatic Surface Mesh functionality and save your new surface mesh.

Surface Mesh Edit

One of the beautiful things of the *Automatic Surface Mesh* (in addition to creating your surface mesh with minimum effort) is that it allows you to edit your final surface grid if needed using the wealth of tools available in Pointwise.
In this section, we will edit the generated surface grid and add cell stretching (i.e. anisotropic cells) on the blade surfaces near the trailing edges. This will resolve the *Area Ratio* problem described at the end of the previous section (*Figure 4*).
Let us start by changing the display settings of the database and domain entities to better visualize the new surface grid.

1. Select the *View* menu option (upper side of the Pointwise interface).
2. Select *Show Database* from the pull-down list.
   This effectively turns off the display of database entities to reduce clutter in the Display window.
3. Select the *List* tab.
4. Click the *Domains* label on the list.
   This selects all the domains in your mesh. You should see the domains highlighted in white in the Display window.
   If your domains are not selected, please check that your Domains selection mask is turned on.
5. Select the *View* menu option (upper side of the Pointwise interface).
6. Select *Attributes* from the pull-down list.
7. In the *Display* frame, select the fourth *Fill* option: *Hidden line* (⬚).
   Note that this display option is also available in the Attributes toolbar for easy access.
8. Click *Update Entity Display.*
9. *Close*

Now that the *Display* window is free from clutter and we have a nice view of the surface grid, let us add stretching to the blade domains near the trailing edges.

1. Zoom close to one of the quadcopter blades.
2. Note that, among others, you have an upper and lower surface domain.
3. Place your mouse pointer over the upper surface domains and click the left mouse button.
   Placing your mouse pointer over the surface domain should cause it to be rendered in a salmon color. Once highlighted, clicking the left mouse button effectively selects the upper domain.
4. Tap the *Space* bar and note that the lower surface domain will be highlighted in salmon color as well.
   Tapping the Space bar allows Pointwise to loop over all the entities that could be selected. This is based on the current state of the selection masks and the current position of the mouse pointer.

5. Press and hold the *Control* key.
   This enables the selection of multiple entities.
6. Select the lower surface domain for the same blade.
   At this point, you should have the upper and lower surface domains selected.
7. Repeat Steps 1 thorough 5 and select the upper and lower surface domains of the other three blades.
   At this point, you should have eight surface domains selected. Two on each blade.
8. Select the *Grid* menu option (upper side of the Pointwise interface).
9. Select the *Solve* option from the pull-down list.
10. Select the *Boundary Conditions* tab.
    In this panel, you will find all the needed tools to define the desired T-Rex boundary conditions.
11. Click *New* to create a new T-Rex boundary condition.
12. Double click *bc-3* in the *Name* column.
13. Enter *TEs* and press *Enter.*
14. Double click *Off* in the *Type* column.
15. Select *Adjacent Grid* from the pull-down list.
16. Zoom in close to the trailing edge of one of the blades.
17. Place the mouse pointer over the edges connecting the trailing edge domain and the upper surface domain.
18. Click the left mouse button to select the edge.
19. Place the mouse pointer over the edges connecting the trailing edge domain and the lower surface domain.
20. Press and hold the *Control* key.
    This enables the selection of multiple entities.
21. Click the left mouse button to select the edge.
    At this point you should have both edges on the blade selected.
22. Repeat Steps 16 through 21 and select the same edges on the other three blades.
    At this point you should have eight edges selected. Two on each blade.
23. Click the *Set* checkbox in the new row in the *Boundary Conditions* table.
24. Note that the number of edges in the new boundary condition (8) is indicated in the *#* column.
25. Select the *Solve* tab.
26. Click *Refine.*
    This refines the selected surface domains (i.e. upper and lower domains on each blade) and applies the new T-Rex boundary condition.
27. Click *OK.*
    This closes the Boundary Conditions panel and saves your changes.

Your surface grid should look similar to what is shown in *Figure 5*. In this image, we show a clear comparison of the upper domain on one of the blades before and after stretching was applied.
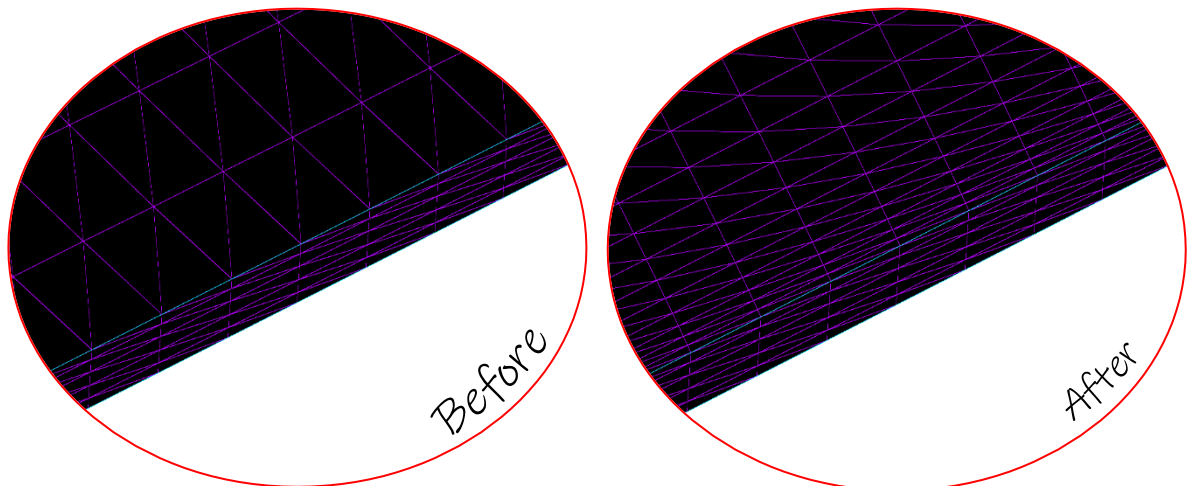
Figure 5: Stretching applied to the upper and lower surface domains of each balde in the trailing edge region. On the left side, we show one of these domains without stretching. On the right side we show the same domain with stretching.

Surface Mesh Examination

In order to make sure our changes to the surface grid generated using the *Automatic Surface Mesh* functionality resolved the *Area Ratio* issue shown in *Figure 4*, let us use the tools available in the *Examine* functionality to check the quality of the final surface mesh.

1. As done before, select all the surface domains.
2. Select the *Examine* menu option.
3. Select the *Area Ratio* metric from the pull-down list.
4. Note in the color bar that the maximum value of the *Area Ratio* in now around 3. This is certainly below the defined threshold value for this metric (i.e. 10).
5. At this point, we would recommend you review the available options in the *Examine* functionality to become familiar with it. For additional information, please review this section of the *Pointwise User Manual*.
6. Click *Close*.

Volume Mesh Creation

Now that the surface mesh is complete and examined, we are ready to create the volume grid. To this end, we will use the *Automatic Volume Mesh* functionality which will allow you to quickly assemble a farfield block in an automated manner and with minimum effort.

1. Select all the surface domains.
2. Select the *Create* menu option.
3. Select the *Automatic Volume Mesh* option from the pull-down list.
4. Select the *Create Volume Mesh Boundaries* option.
   This option automatically defines symmetry planes and farfield extents for external volume grids.

5. Make sure the *Global* tab is currently selected.
6. Select the following options:
   a. *Type*: *Voxel* (i.e. second option)
      The new volume grid will be of Voxel type.

   b. *Shape*: *Box* (i.e. first option)
      The farfield block will have a box shape.

   c. *Center: Margins* (i.e. first option)
      The extents of the box farfield will be computed relative to the faces of the extents box of the surface grid.
7. Specify the following margins to define the size of the farfield block:
   a. L_1: 500
      Note that L_2 will be set to the same value automatically.

   b. W_1: 300
   c. H_1: 500
      Note that H_2 will be set to the same value automatically.

Once the above parameters are specified, the geometrical shape and size of the farfield block should look similar to *Figure 6*.
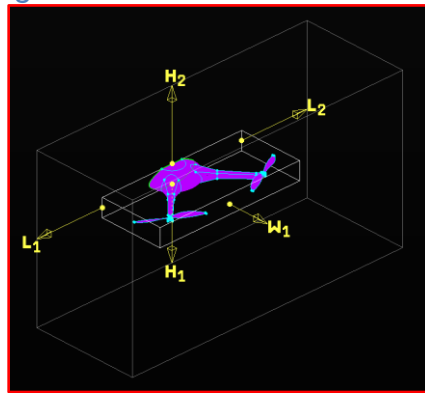

Figure 6: The specified parameters define the geometrical shape and size of the farfield block.

Since we are interested in creating a viscous volume grid, we will need to add anisotropic cells in the boundary layer region (i.e. volume cell stretching).

8. Select *Geometric Growth* from the *Boundary Layer* pull-down list.
9. Enter *0. 001* in the *Wall Normal Spacing* entry field.
10. Select the *Boundary Conditions* tab.
11. Note that the needed [T-Rex boundary conditions](#) have been defined automatically.
12. Select the *Advanced* tab.
13. Under *Symmetry Planes*, note that a *+X* symmetry has been automatically recognized.
14. Also note that the symmetry edge on the surface grid is rendered green (*Figure 7*).
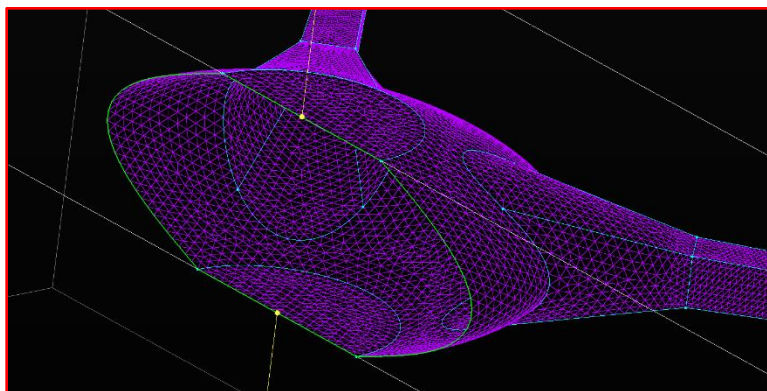
Figure 7: The symmetry edge in the surface grid is automatically recognized and rendered in green for easy visualization.

15. Select the *Global* tab.
16. Click *Create Volume Mesh.*

The new volume grid has been created. However, note that at this point the new block is empty (i.e. it doesn't have interior cells).

17. Check on the *Populate Volume Mesh on Command Completion* checkbox.
    This will ensure that the new volume grid gets initialized (i.e. populated with cells) as soon as you click the OK button to exit the Automatic Volume Mesh functionality.

18. Click *OK.*
    This exits the Automatic Volume Mesh functionality and saves your changes. Note that, as mentioned in Step 17 above, the volume grid will be initialized as soon as this functionality exits. The initialization process can take a couple of minutes depending on your machine.

The new volume grid should look similar to the one shown in *Figure 8*.
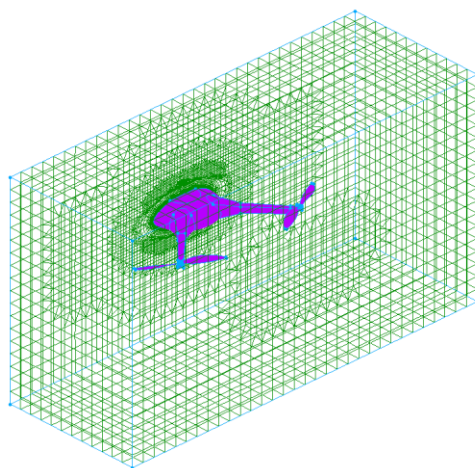

Figure 8: Final voxel volume grid assembled using the Automatic Volume Mesh functionality with minimum input from the user.

Volume Mesh Smoothing

Note that an extrema value of ~175 degrees for the *Maximum Included Angle* examine function (shown above) is suitable for most commercial CFD solvers. That being said, if your solver has more restrictive requirements, you can use the *Smoothing* tool provided by the [unstructured solver](#) to smooth the volume mesh and reduce the obtained extrema value.

1. Select the block.
2. Select the *Grid* menu option.
3. Select the *Solve* option from the pull-down list.
4. Select the *Attributes* tab.
5. Check on the *Smoothing* checkbox.
6. Enter *170.0* in the *Cost Angle Threshold* entry field.
   The Cost Angle is an alternate form of the Maximum Included Angle ([Smoothing](#) section of the Pointwise User Manal). This parameter set the angle to determine which volume cells in the mesh should receive smoothing.
7. Select the *Solve* tab.
8. Enter *100* in the *Iterations* entry field.
   This parameter specifies the desired number of smoothing iterations.
9. Click *Smooth*.
   This starts the Weighted Condition Number smoother which will look to smooth the volume cells with a Cost Angle larger than the value specified in Step 6 and will run the number of iterations specified in Step 8.

It is important to note that the *Examine* functionality used previously can be accessed directly from the unstructured solver panel.

10. Select the *Max. Included Angle* option from the *Functions* pull-down list.
11. Click *Examine*.
12. In the *Extrema* frame, note the extrema value for the *Maximum Included Angle* function. The value should have decreased considerably from its initial value close to 175 degrees.
13. Click *Close*.
    This closes the Examine functionality and puts you back in the Unstructured solver panel.

If the final grid is appropriate for your CAE solver, you can simply click *OK.* However, if you would like to further smooth your grid, you can simply run another pass of the Weighted Condition Number smoother.

14. Click *OK*.
    This closes the unstructured solver panel and saves your changes.

Set CAE Boundary Conditions

Prior to [exporting](#) your finished grid, set up the [boundary conditions](#) that you plan to use in the currently selected CAE solver ([OMNIS](#) in this case). Setting these boundary conditions now will make it easier for you to identify regions in your grid once it is imported into your solver.

1. Select the *CAE* menu option.
2. Select the *Set Boundary Conditions* option from the pull-down list.
3. Click *New*.
   This creates a new CAE boundary condition.
4. Double-click the *Name* field of the new boundary condition (*bc-2*).
5. Enter *Farfield*.
6. Double-click the *CAE Type* field of the *Farfield* boundary condition.
7. Select *Farfield*.
8. Select the five domains defining the farfield box (do not select the symmetry domain).
9. Check the *Farfield* boundary condition.
   This assigns the selected domains to Farfield boundary condition.
10. Click *New*.
11. Double-click the *Name* field of the new boundary condition (*bc-3*).
12. Enter *Quadcopter*.
13. Double-click the *CAE Type* field of the *Quadcopter* boundary condition.
14. Select *Wall Viscous*.
15. Select all the surface domains on the quadcopter.
16. Check the *Quadcopter* boundary condition.
    This assigns the selected domains to Quadcopter boundary condition.
17. Click *New*.
18. Double-click the *Name* field of the new boundary condition (*bc-4*).
19. Enter *Symmetry*.
20. Double-click the *CAE Type* field of the *Symmetry* boundary condition.
21. Select *Symmetry Plane*.
22. Select the symmetry domain.
23. Check the *Symmetry* boundary condition.
    This assigns the selected domain to Symmetry boundary condition.
24. Click *Close*.
    This closes the CAE Boundary Conditions panel and saves your changes.

You have now assigned domains to boundary conditions for export to [OMNIS](#).

Save Project

Save your final mesh to your project file now so you can easily access your mesh in Pointwise while retaining all previously used layer and display settings.

1. Select the *File* menu option.
2. Select the *Save As* option from the pull-down list.
3. Navigate to your preferred save folder.
4. Enter *quadcopter-ASM-Final.pw* in the *File name* field.
5. Click *Save*.

CAE Export

You can now export your mesh to [OMNIS](#) and start generating solutions for your application.

1. Select the block
2. Select the *File* menu option.
3. Select the *Export* option from the pull-down list.
4. Select the *CAE* option from the secondary pull-down list.
5. Specify a filename.
6. Click *Save*.
7. Click *OK.*

Support

Cadence Learning and Support Portal provides access to support resources, including an extensive knowledge base, access to software updates for Cadence products, and the ability to interact with Cadence Customer Support. Visit https://support.cadence.com.

Feedback

Email comments, questions, and suggestions to content_feedback@cadence.com.