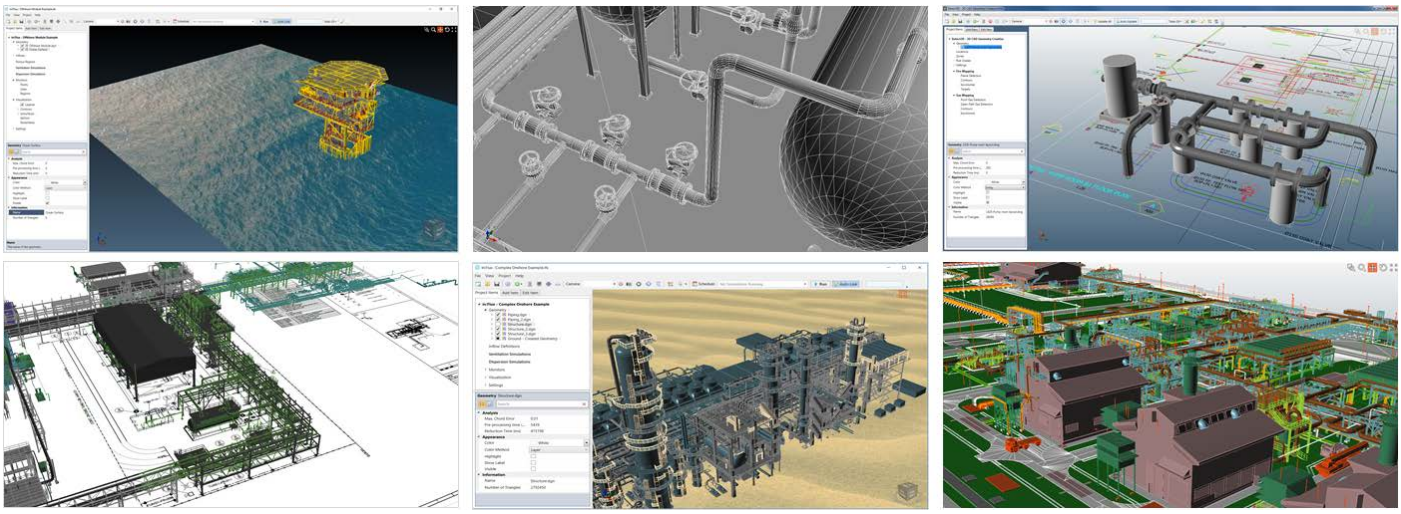
CAD Geometry Guide



This section of the guide will go over the varying CAD geometry import formats and capabilities of Detect3D and in:Flux. Here you can search through topics below or on the left panel, click the topic to open its associated page.

Insight Numerics provides CAD conversion and CAD import services free of charge for license holders. Please contact info@insightnumerics.com with any questions about this service.

CAD Import Guide

Both Detect3D and in:Flux allow for a variety of CAD models to be imported. This section is intended to help users with the varying ways to import and manage CAD models for analysis.

There is no limitation on the size or complexity of file which can be imported. The main factor in this process is the amount of memory available on your machine - the more complex the file, the more memory needed.

If you need help with importing or converting a file, Insight Numerics provides free conversion services for license holders. Please contact info@insightnumerics.com with any questions.

Accepted File Types

Detect3D and in:Flux can import any of the 8 following CAD geometry formats below.

Best options:
DGN: Microstation file format, PDMS can export to DGN. SmartPlant can also export to DGN with the Interop Publisher package.
DWG: AutoCAD native file format
DWF: format exported from Navisworks Manage or Navisworks Simulate. Best way from converting from NWD or other CAD formats not listed here.

Also accepted:
DXF: AutoCAD file format for non-AutoCAD applications
IGS: a CAD format based on the Initial Graphics Exchange Specification
STEP: a standardized CAD file format under ISO 10303
OBJ: Triangulated file format used for storing a description of the surface of 3D objects, usually has an associated .mtl (material file), that is used for layers. For the layers to be imported correctly the .mtl file should be in the same directory as the .obj file
STL: Triangulated file format, the simplest 3D CAD file which does not have any layers.

IFX: The geometry model from a saved in:Flux project file (.ifx) may be selected when importing CAD models. Doing so will only load the CAD geometry from the ifx file.

D3D: Similar to above, any saved Detect3D file (.d3d) with geometry can be selected when importing CAD to in:Flux. Doing so will only import the CAD geometry from the selected file into the current in:Flux project.

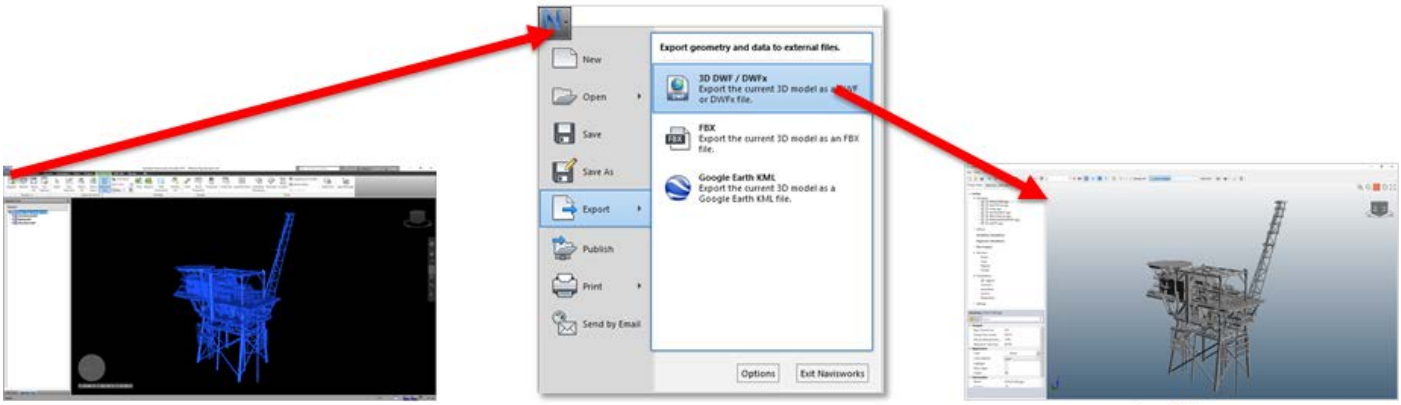
Navisworks and DWF Files

In 2019 Insight Numerics released an update for importing DWF files into Detect3D. This DWF file can be exported from Navisworks Manage or Navisworks Simulate software. Since Navisworks can import a large variety of CAD formats (including NWD and NWC files) it greatly reduces time spent for conversions.

To create a dwf file:
1. Load your CAD model into Navisworks Manage or Navisworks Simulate
2. Click the main menu "N" button on the upper left corner of the screen and choose the *Export* option
3. Select "3D DWF/DWFs" as the format and choose a directory to save the file
4. Once exported, open Detect3D or in:Flux and choose the saved DWF file
5. When the CAD Import Options window appears, ensure that 'meters' is selected - this will match the scaling and coordinates from Navisworks
6. Click the *Import* button

For a video walk though of the above process, enter the following YouTube link into your browser:

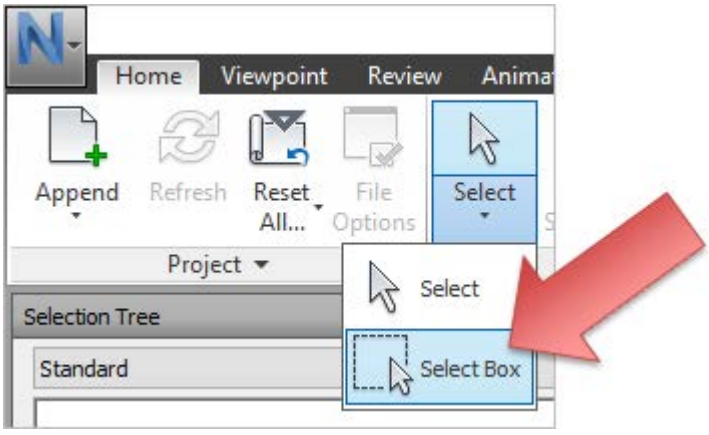
https://www.youtube.com/watch?v=O8vE_W43U70



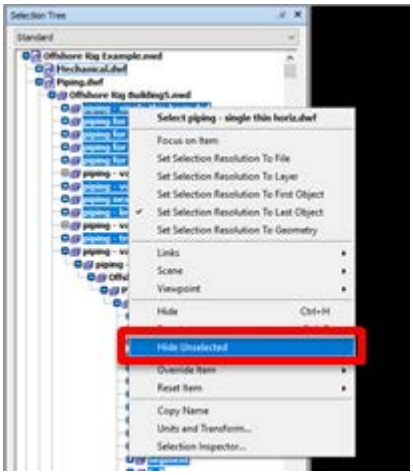
Dividing Up CAD models in Navisworks

On occasion, clients provide the entire site CAD model for F&G mapping analysis. These files can have sites larger than 1 km in length or with multiple complex offshore decks. Depending on your machine, these files may make it difficult for computers to process the file. Fortunately, Navisworks can also be used to help manage these larger complex CAD files.

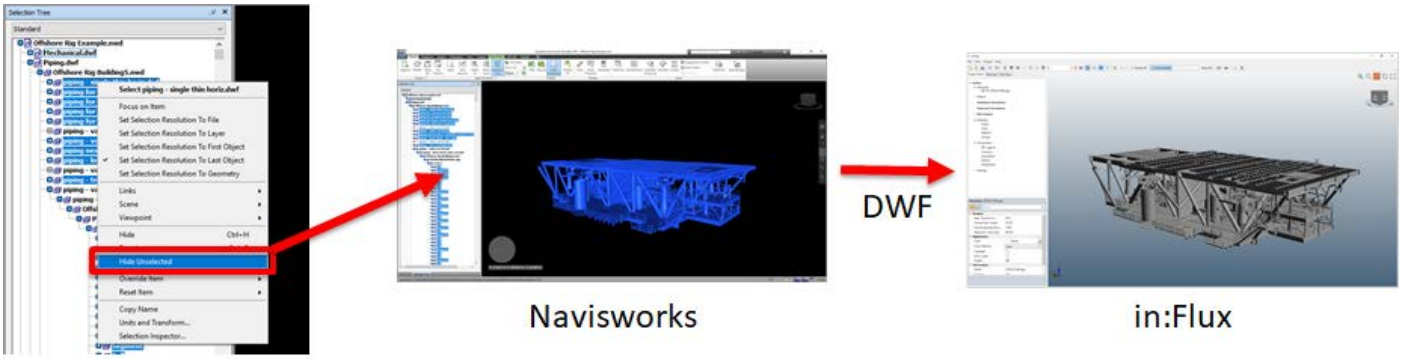
After importing your CAD geometry into Navisworks Manage or Navisworks Simulate, you can select individual layers or equipment pieces from the 'Selection Tree' or highlight sections with the 'Select Box' tool, shown below.



Once desired items have been selected, from the 'Selection Tree' right-click on the blue selected items and choose “Hide Unselected”



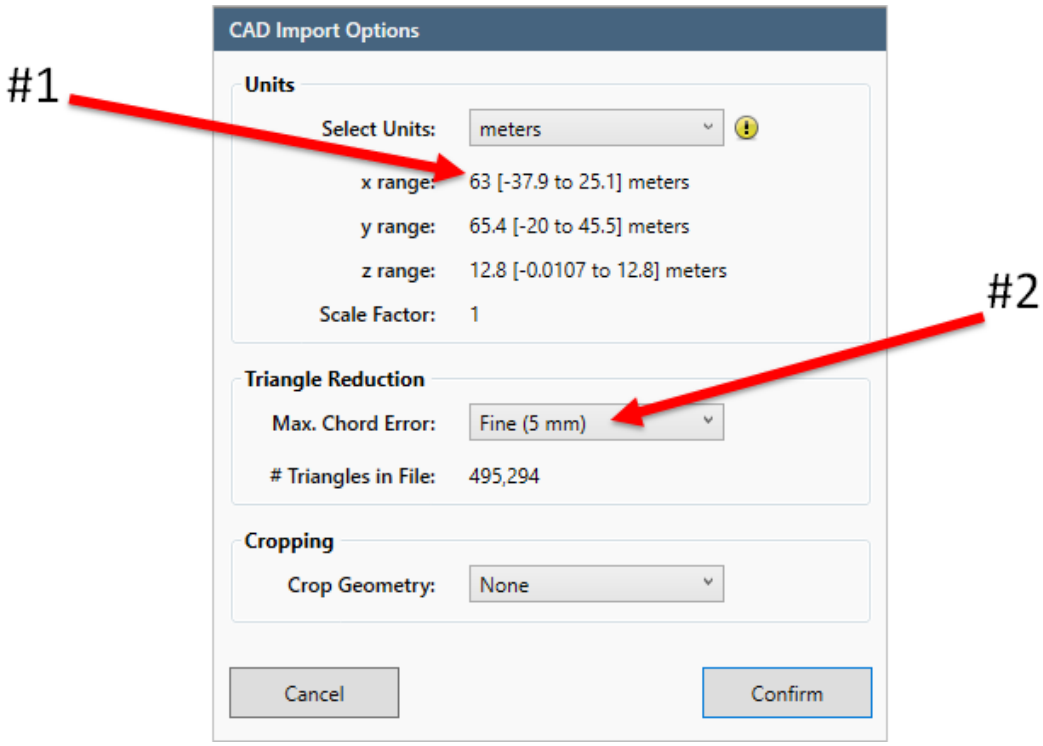
Now export the file to DWF using the same steps above, this time only the selected items will be exported. This process can be repeated by un-hiding and re-hiding CAD pieces.



A follow-up video to the one above, showing the 'Select Box' cropping process can be watched on YouTube here: https://www.youtube.com/watch?v=ezjmztnISQY

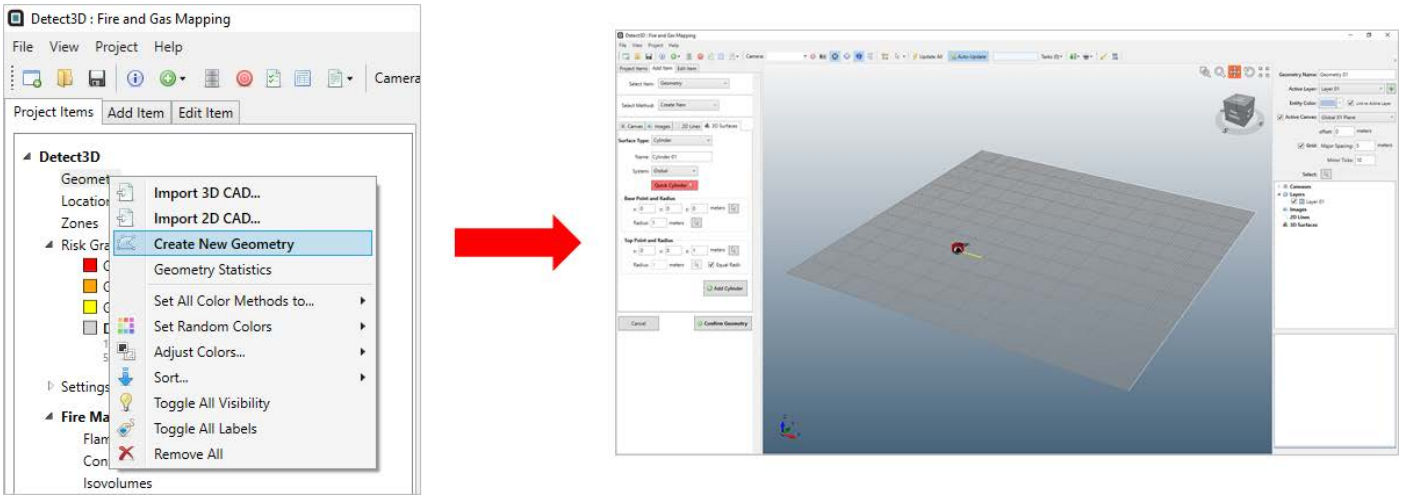
CAD Import Checklist

1. Check dimensions listed on the CAD Import Options Window:
    a. Changing the units will scale the file, make sure the dimensions match what you would expect for your facility
2. Triangle reduction listed on the CAD Import Options Window:
    a. Leave at default option or reduce if you think necessary
3. Once imported:
    a. Check scaling with a known dimension from plot plan - sometimes human figures are left in the file, use the measure tool and ensure their height is near 2 meters. Railing and stair widths are also a good reference for scaling
    b. Check units - use Detect3D or in:Flux coordinate system or the CAD models coordinate system. For files with coordinates taken from latitude or longitude (placing the origin far away from the CAD model) it is advised to translate the file near to the origin and use the software coordinate system. Once the analysis is complete you may then change the exported leak location or monitor coordinates to the site coordinates using the excel export options
    c. Rotate model to correct 'platform or site north' position



CAD Creation Tool Guide

The create geometry tool within Detect3D and in:Flux can be accessed via the add items tab or by right clicking the Geometry header in the Project Items Tab and selecting *Create New Geometry.* This will open a new user interface for editing and creating CAD geometry. For step-by-step instructions for building CAD geometry upon an imported image, see the YouTube links below or Detect3D Tutorial 11.



There are 4 tabs on the left panel of create geometry tool which categorize the varying capabilities.

- **Canvas -** This tab will allow you to define a plane on which you can then draw and create objects on. Canvases need to be defined at the height in which you desire objects.
- **Images -** Allows you to import images and orient/scale them appropriately for your project. The images can then be used to aide in creating geometries or for visualization purposes.
- **2D Lines -** Here you can create 2D shapes (polylines, circles, arcs, etc) which can be used when defining 3D surfaces. Most commonly the polyline is used to draw/outline walls of a facility then extruded in the 3D surfaces tab.
- **3D Surfaces -** This tab allows you to add 3D shapes such as cubes, spheres, and cylinders as well as create 3D shapes from 2D lines which you have already defined. The 'Quick' buttons allow for shapes to be made with just 3 clicks of the mouse.

In addition to Detect3D's Tutorial 11, please have a look at our YouTube Videos going over building a CAD geometry from an example plot plan. The walk through is in 2 parts and has no audio but provides a detailed explanation of how to go about building CAD models within in:Flux CFD or Detect3D.

CAD Creation Tool - Part 1: https://youtu.be/A5_dGZ8SanI

CAD Creation Tool - Part 2: https://youtu.be/7nRW3iKPwu8

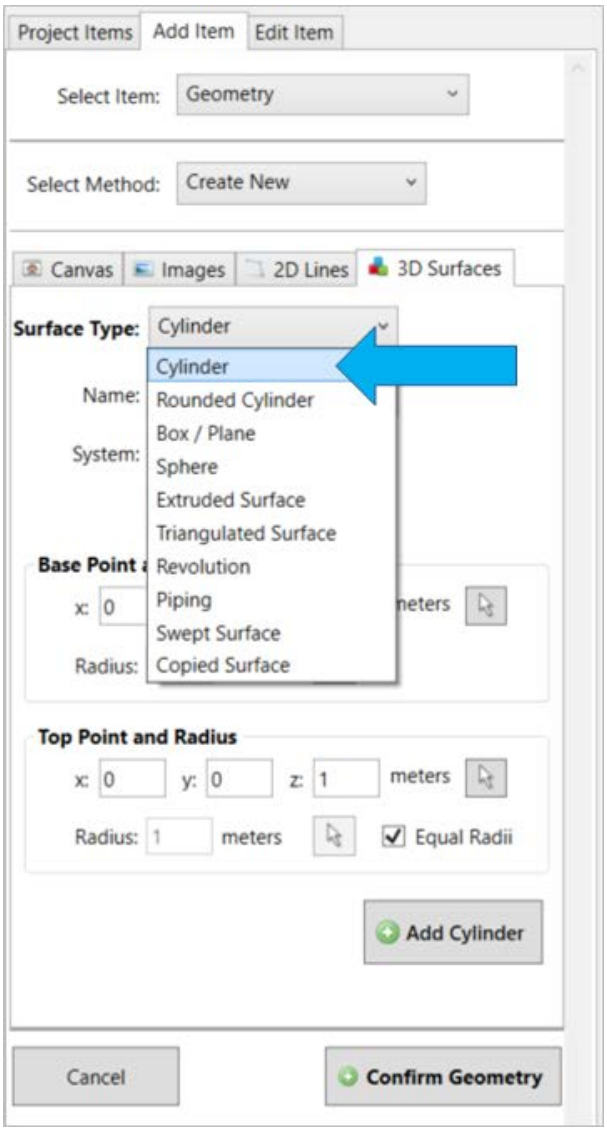
The sections will show how to use the 'Quick' options for some of the available 3D surfaces as well as some additional capabilities for importing images and plot plans:

Quick Cylinder

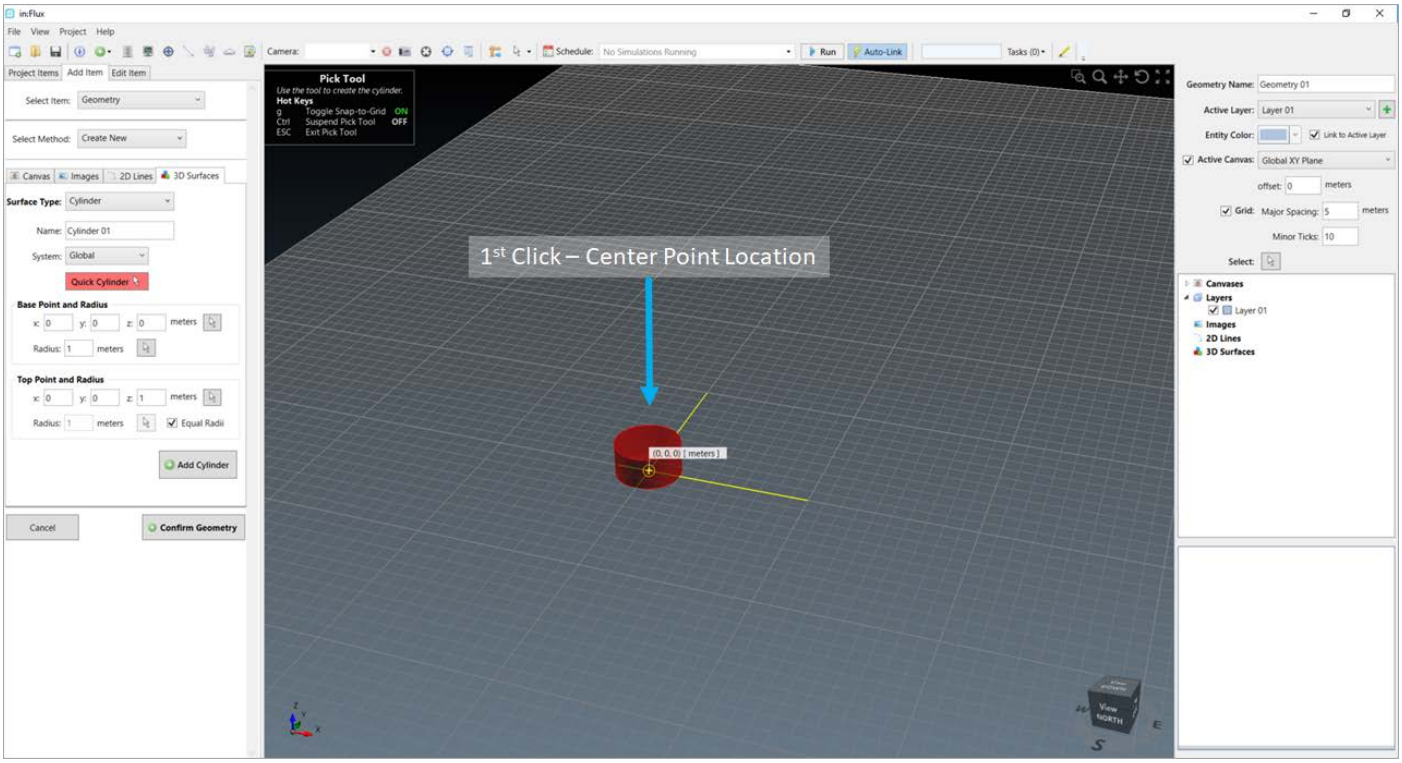
One of the ten options for 3D Surfaces in the CAD Creation Tool is the Cylinder.

A red button in the middle of the panel, 

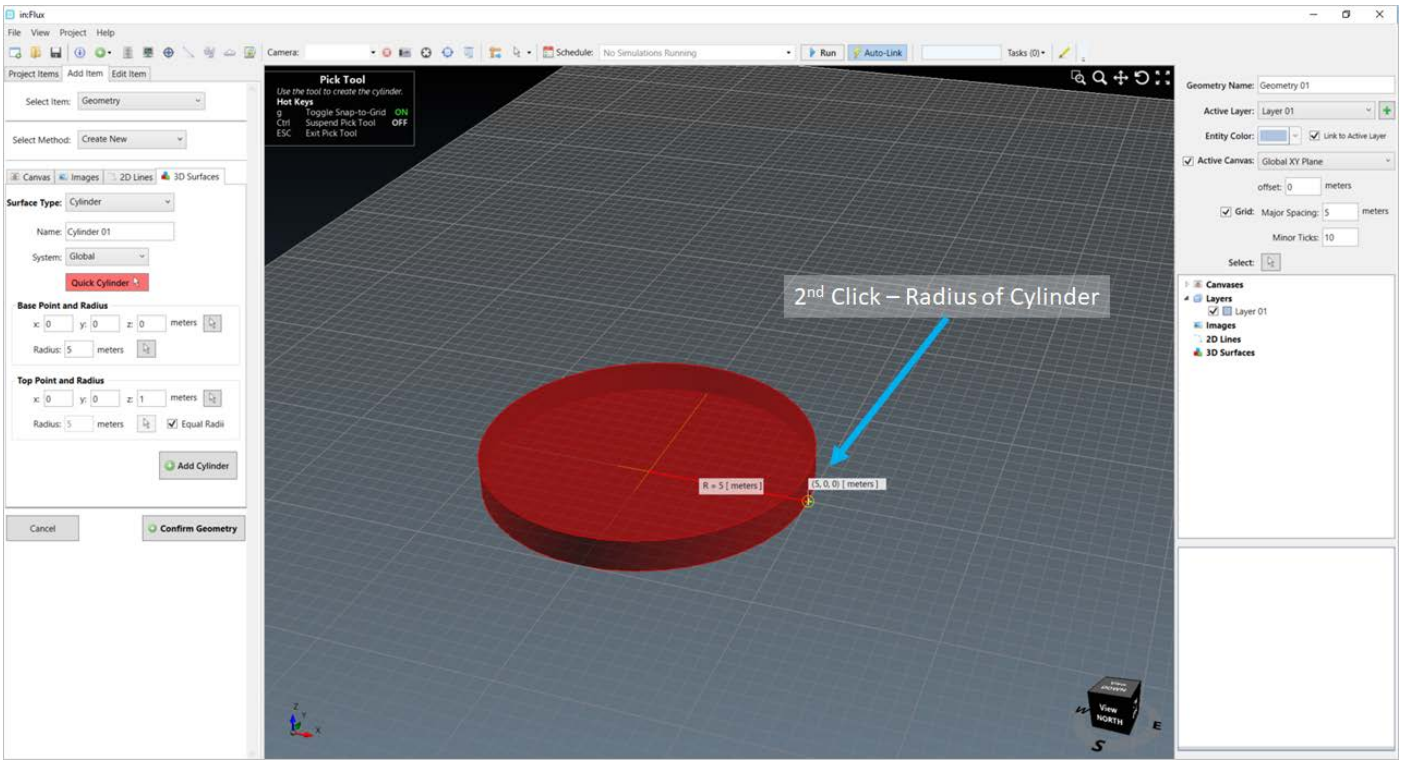
, may be used to create a cylinder with just 3 clicks of a pick tool. The below images describe the process of using the *Quick Cylinder* feature.



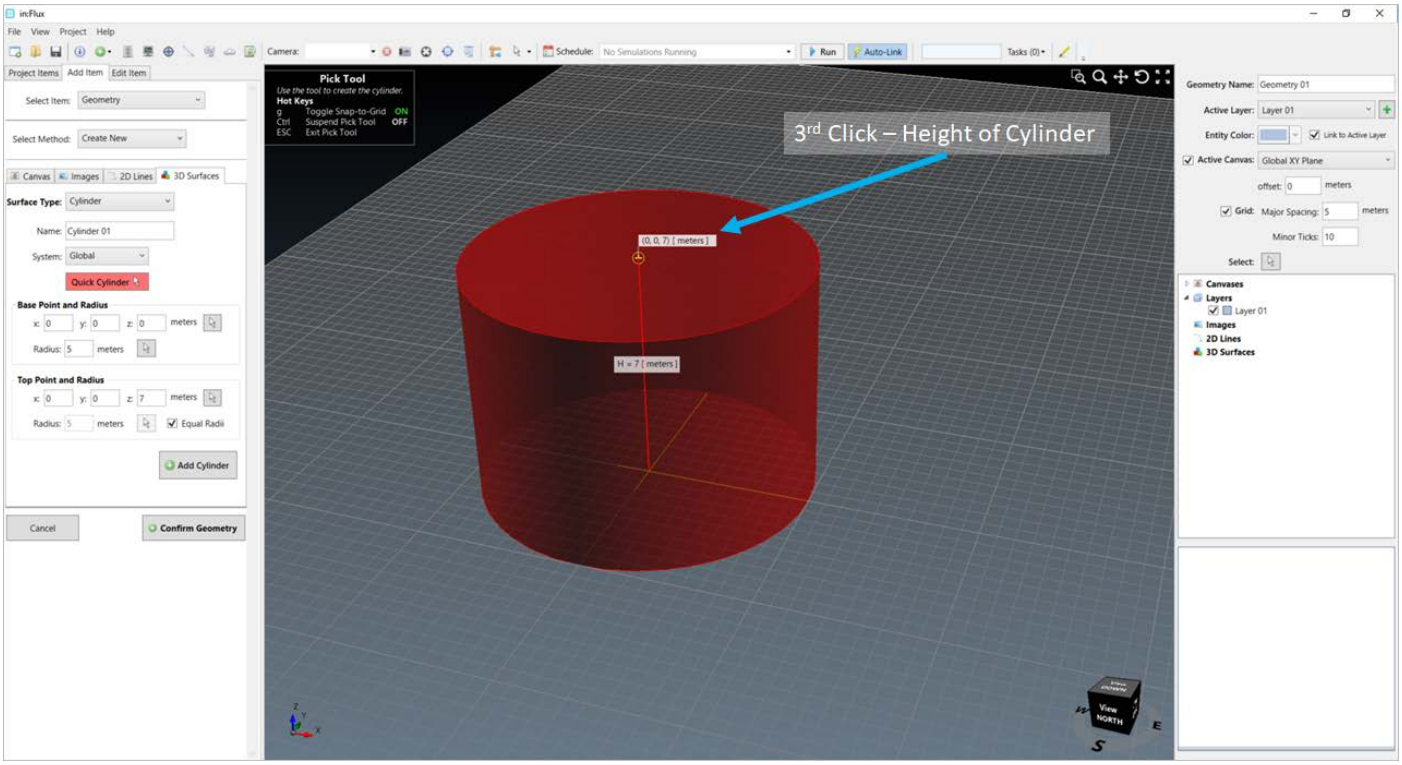
*From the 3D Surfaces tab, select 'Cylinder' from the **Surface Type** dropdown menu*



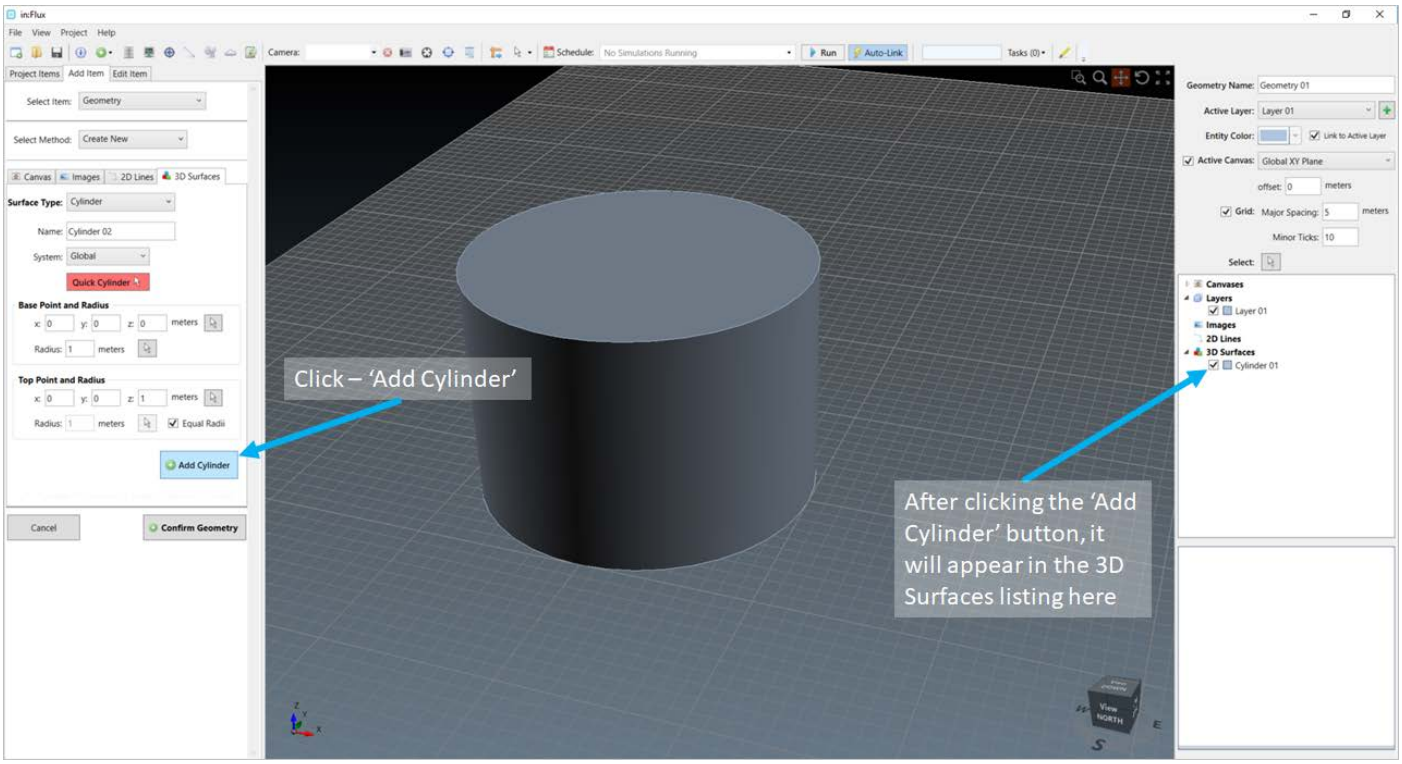
After selecting the red 'Quick Cylinder' button, a pick tool will appear. The first point selected on the canvas will be used as the center of the cylinder



After the first point has been selected, the center will be recorded, and the transparent red cylinder will follow the mouse pointer expanding to its location. The 2nd selected point will correspond to the radius of the cylinder.



With the base created from the 1st two points (center and radius, respectively), the height can then be defined. The transparent red cylinder will follow the mouse pointer until a point is selected.



After the third point is selected, press the 'Add Cylinder' button to add the 3D Surface to the CAD Creation Project. It will be listed on the right-hand panel as indicated above.

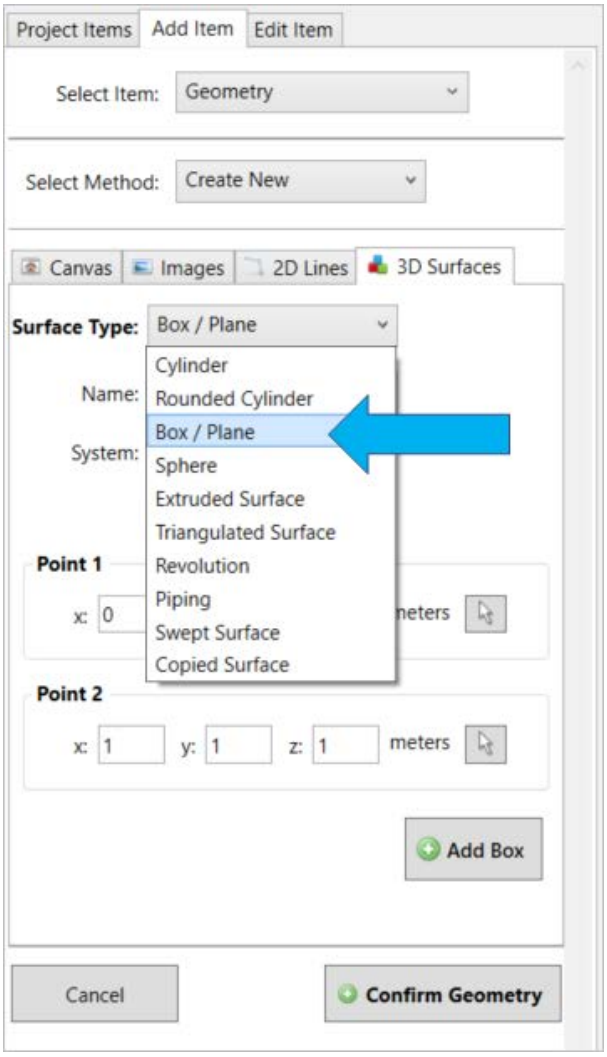
Note that any CAD created in the above interface will not be properly accounted for unless the 

button is clicked. Upon selection, the normal Detect3D or in:Flux interface will appear, and the CAD can be used for future CFD analyses. Any simulations already completed will need to be recalculated to properly include the created CAD geometry.

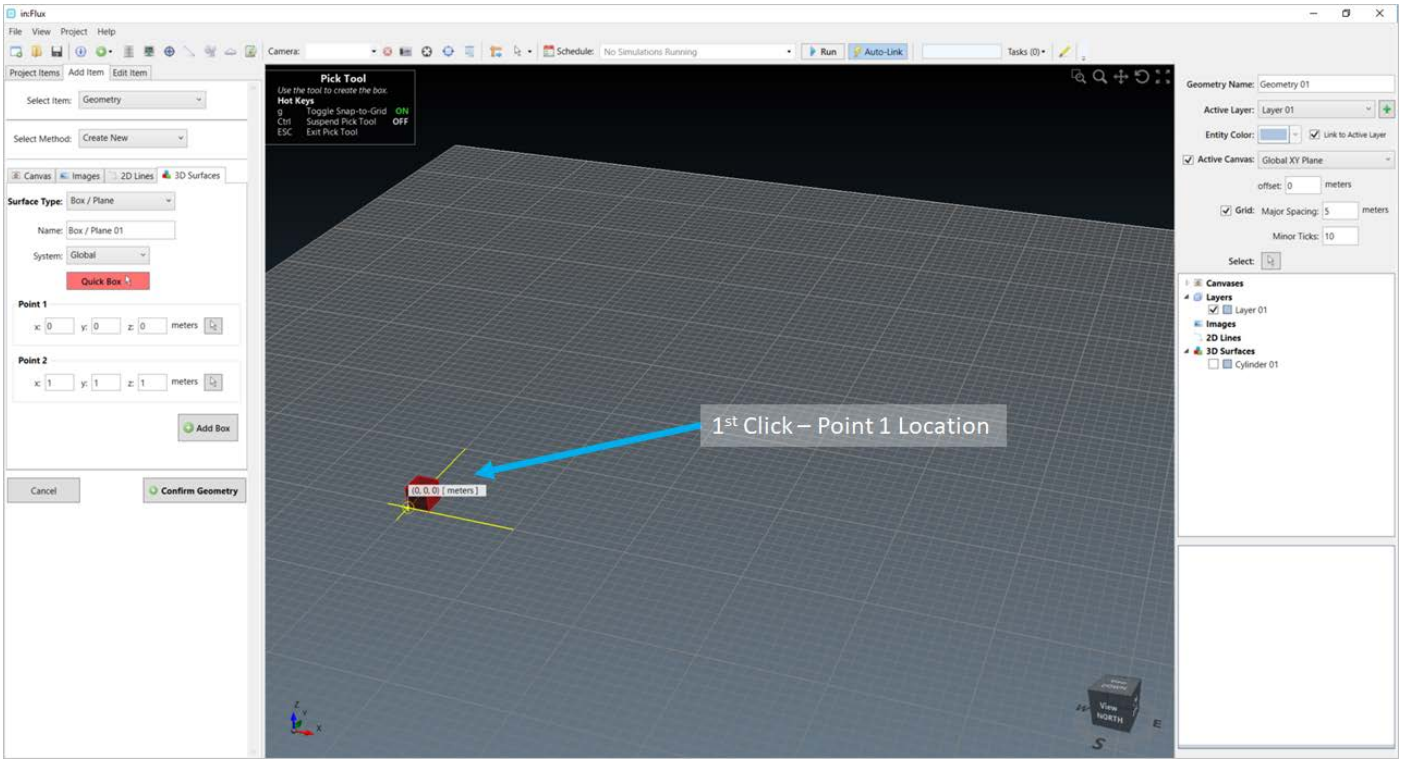
Quick Box or Plane

The *Quick Box* option can be used to create boxes, walls, or ground pieces with ease.

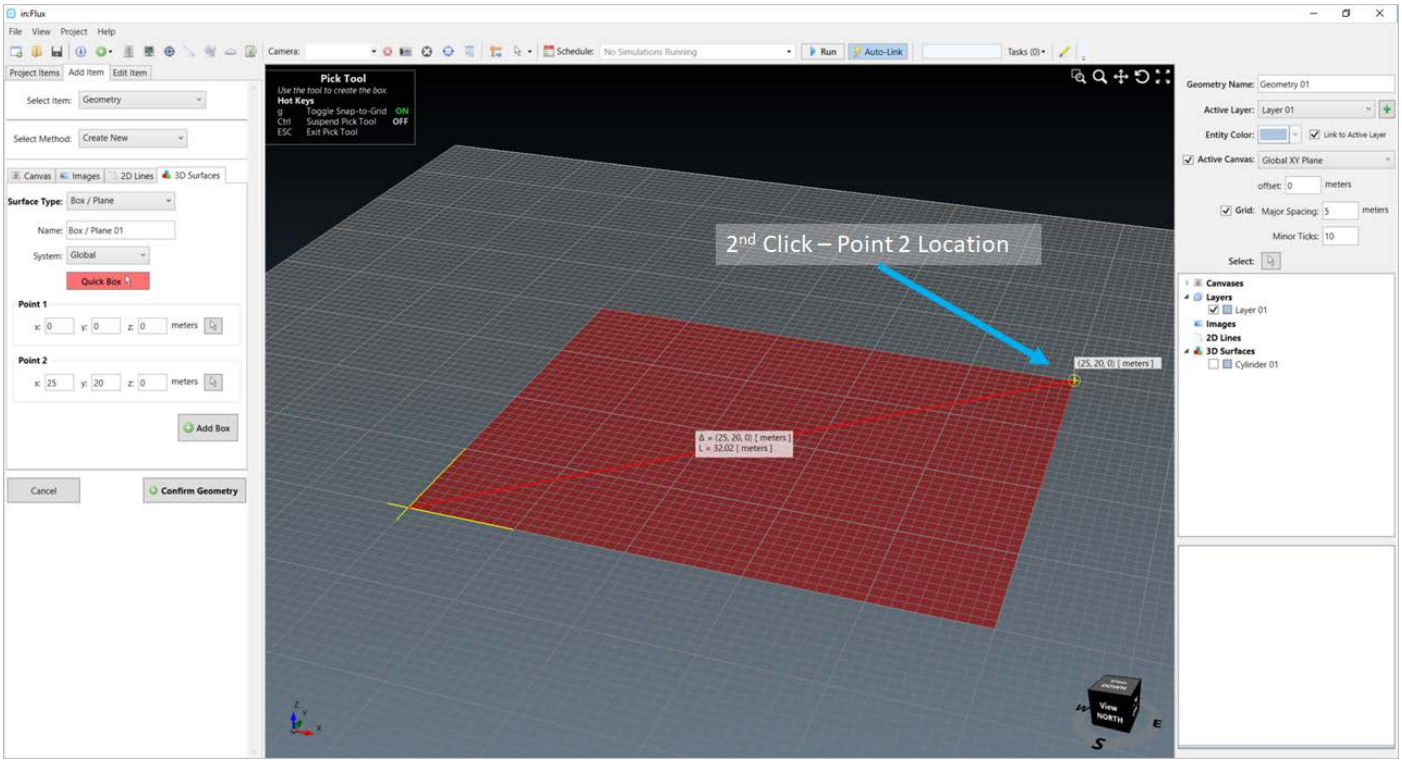
The following images will go over creating a ground object (planar surface) and two walls using the *Quick Box* feature:



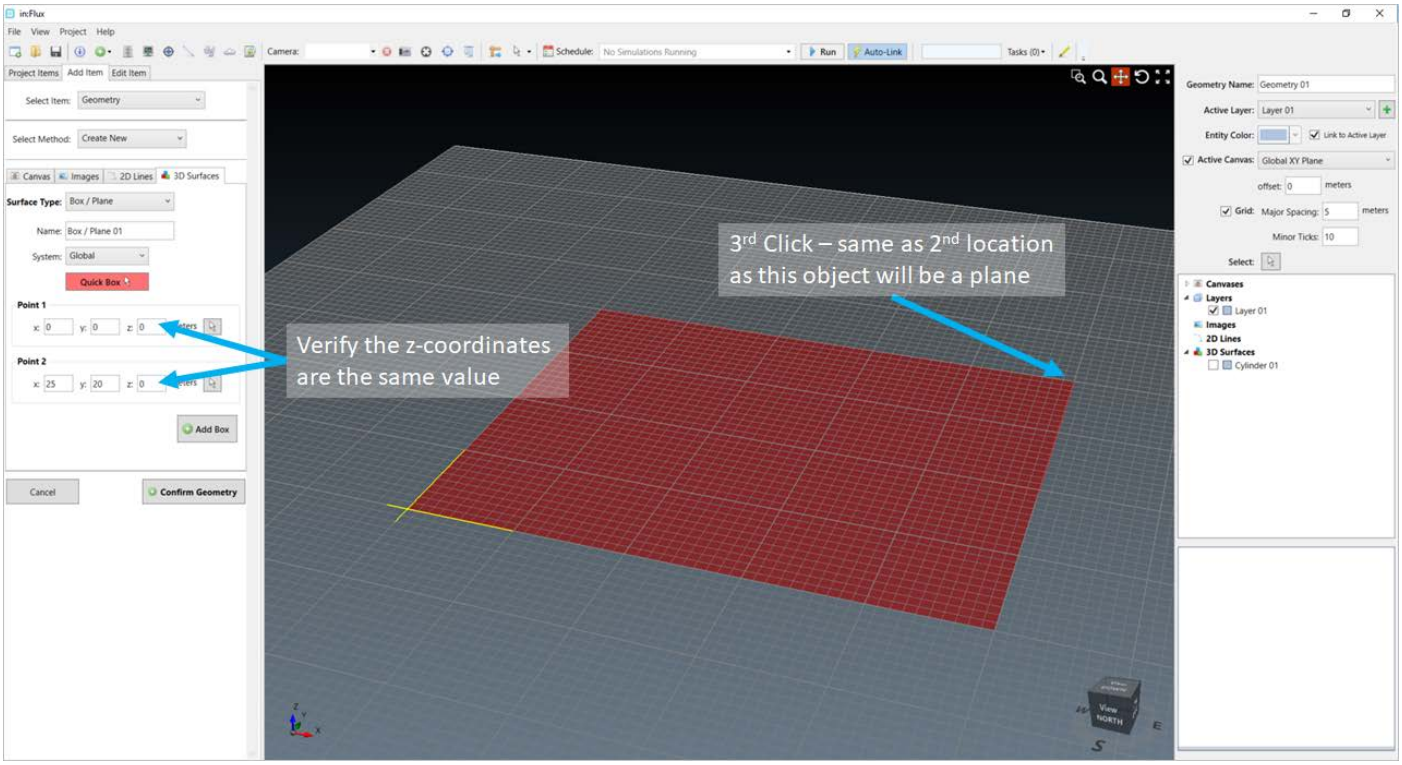
From the 3D Surfaces tab, choose the 'Box / Plane' option



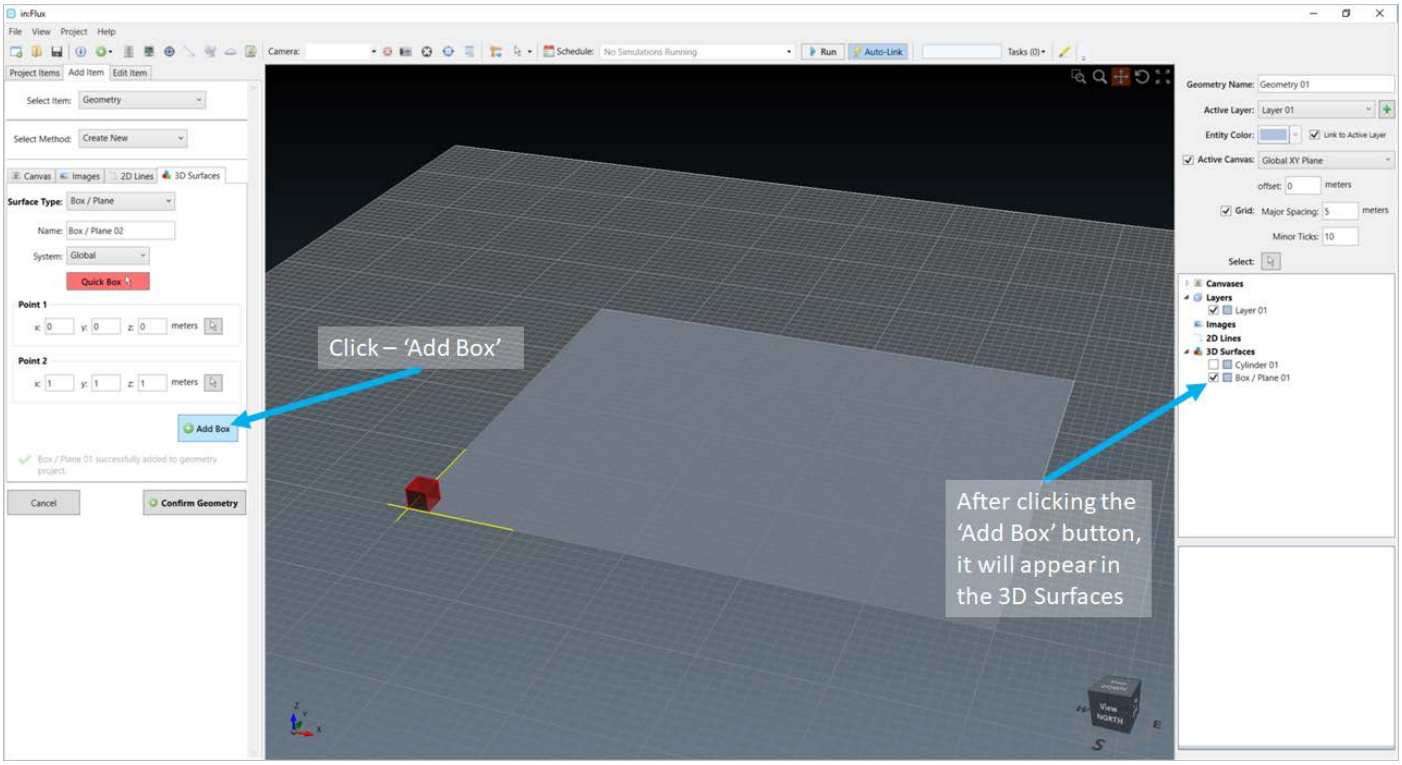
Click the red 'Quick Box' button to activate the tool. The first point selected will act as one of the two outer bounds for the plane



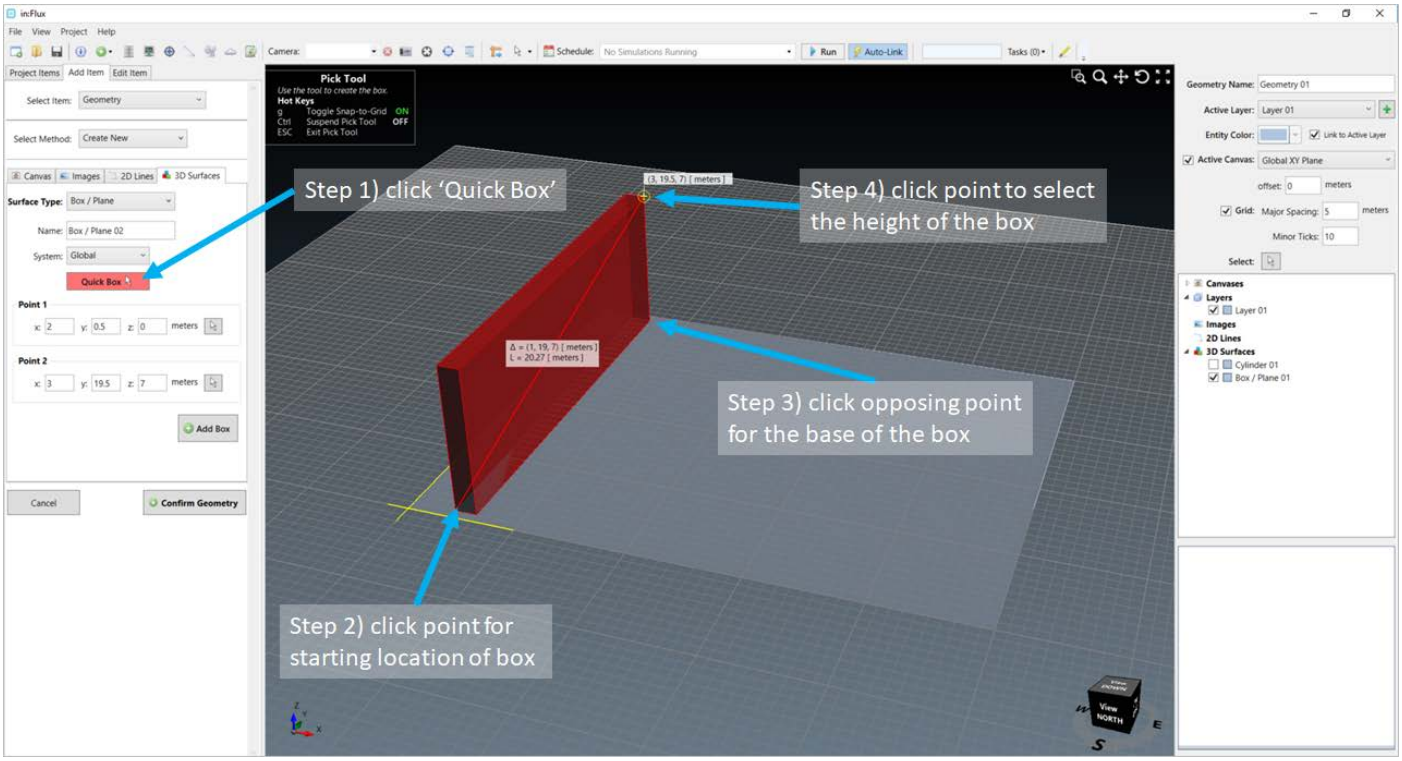
The second point selected will define the extents (length and width) of the box or plane to be defined.



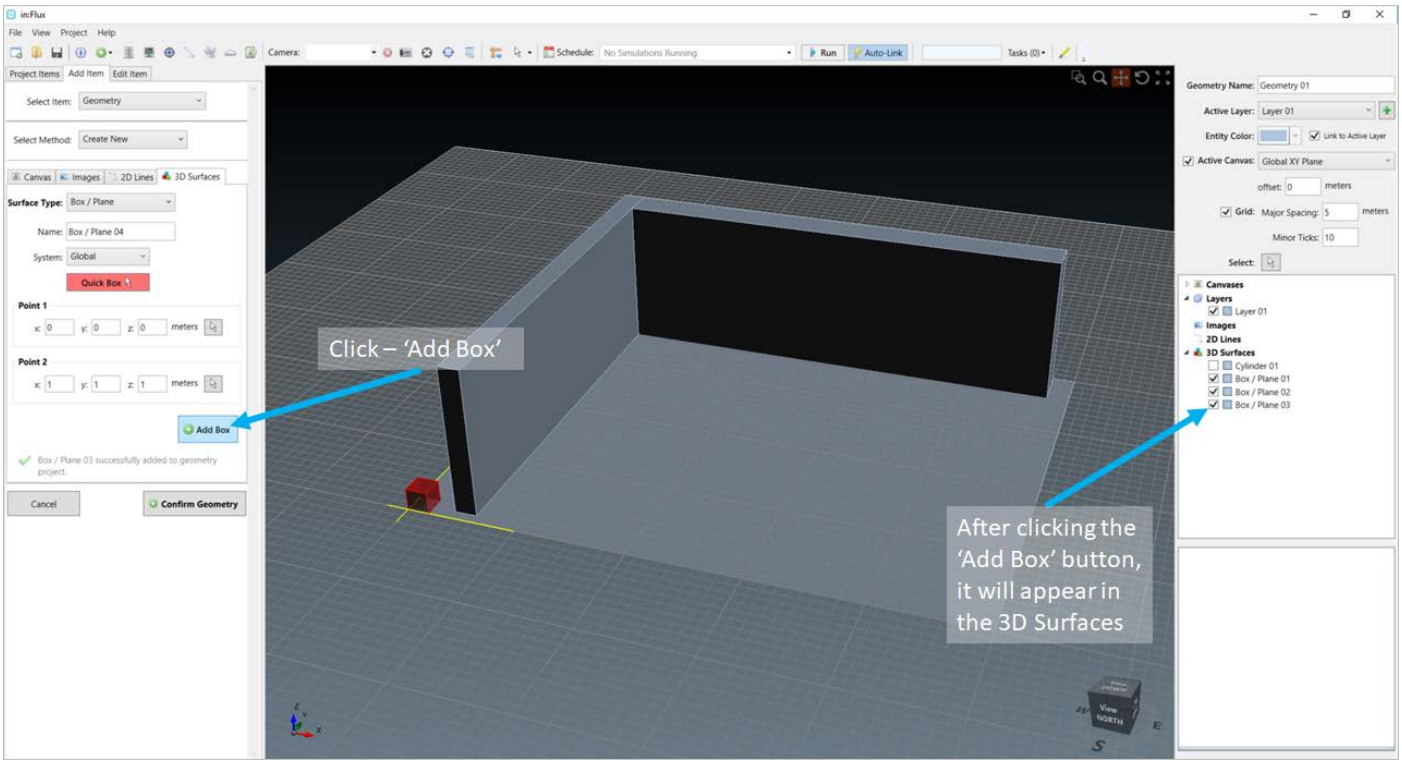
For planes or ground pieces, select the second point again (as the 3rd point) and ensure that the recorded height (z-coordinates) are the same



Click the 'Add Box' button to add the plane to the right-hand panel



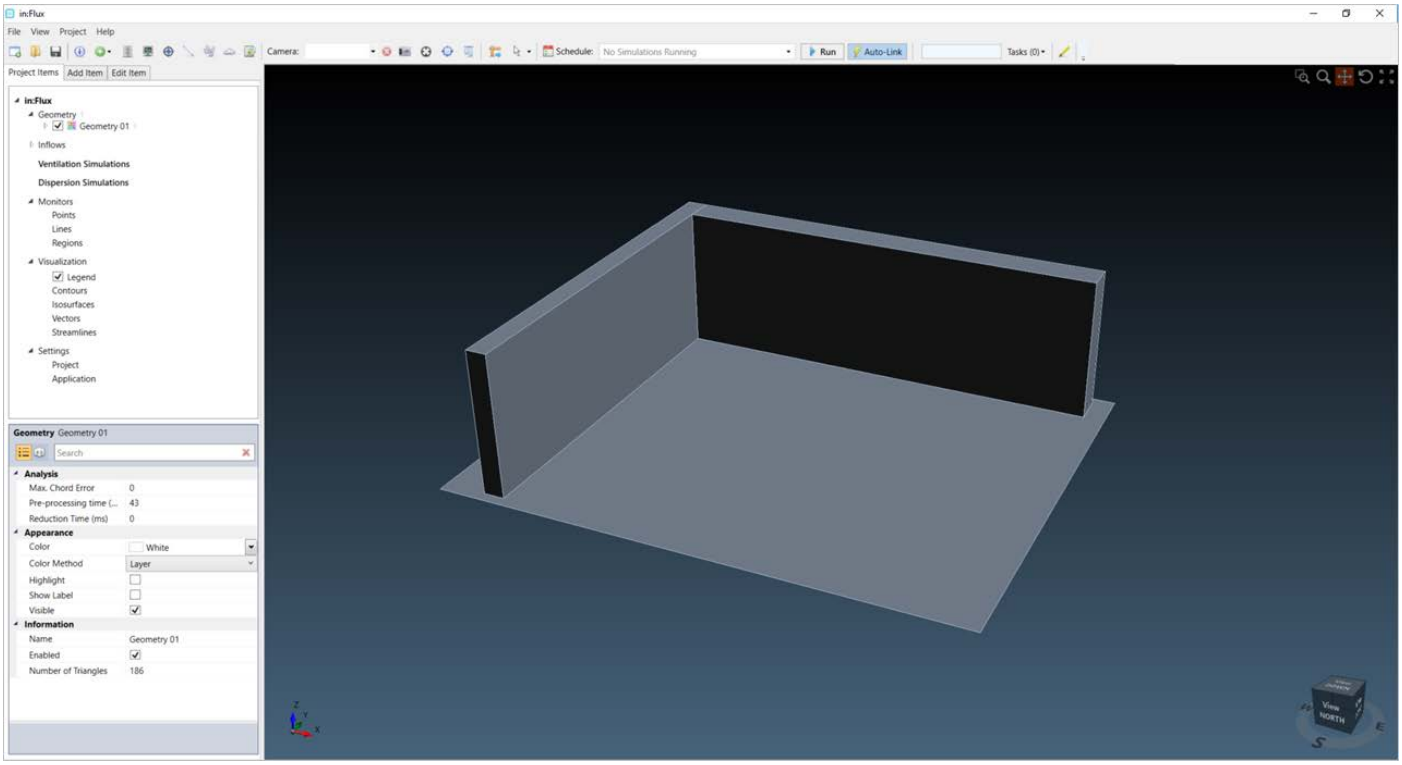
This process can be repeated for creating boxes. The above image shows the process for a wall to be created. Note that the 3rd click here is defining the height of the wall.



Be sure to always click the 'Add Box' button to add the 3D surface to the CAD Creation Project. You may also opt to enter coordinates of the box manually rather than use the 'Quick Box' tool.

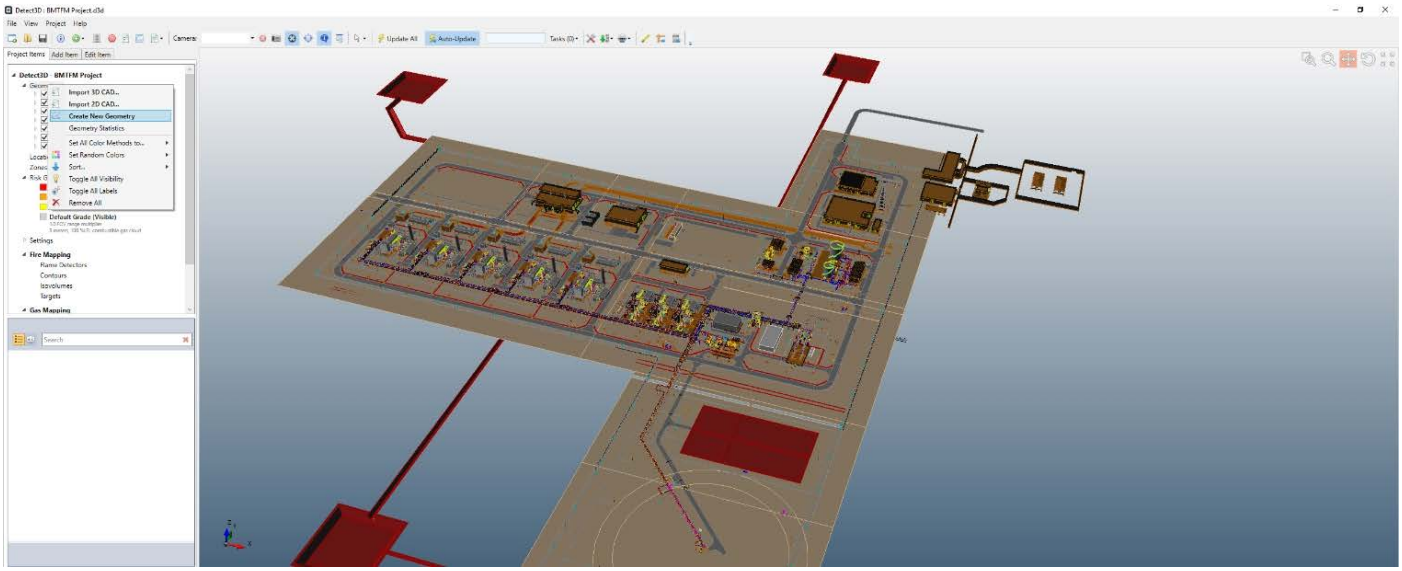
Note that any CAD created in the above interface will not be properly accounted for unless the 

button is clicked. Upon selection, the normal Detect3D or in:Flux interface will appear (as shown below) and the CAD can be used for future analyses. Any simulations or calculations already completed will need to be recalculated to properly include the created CAD geometry.

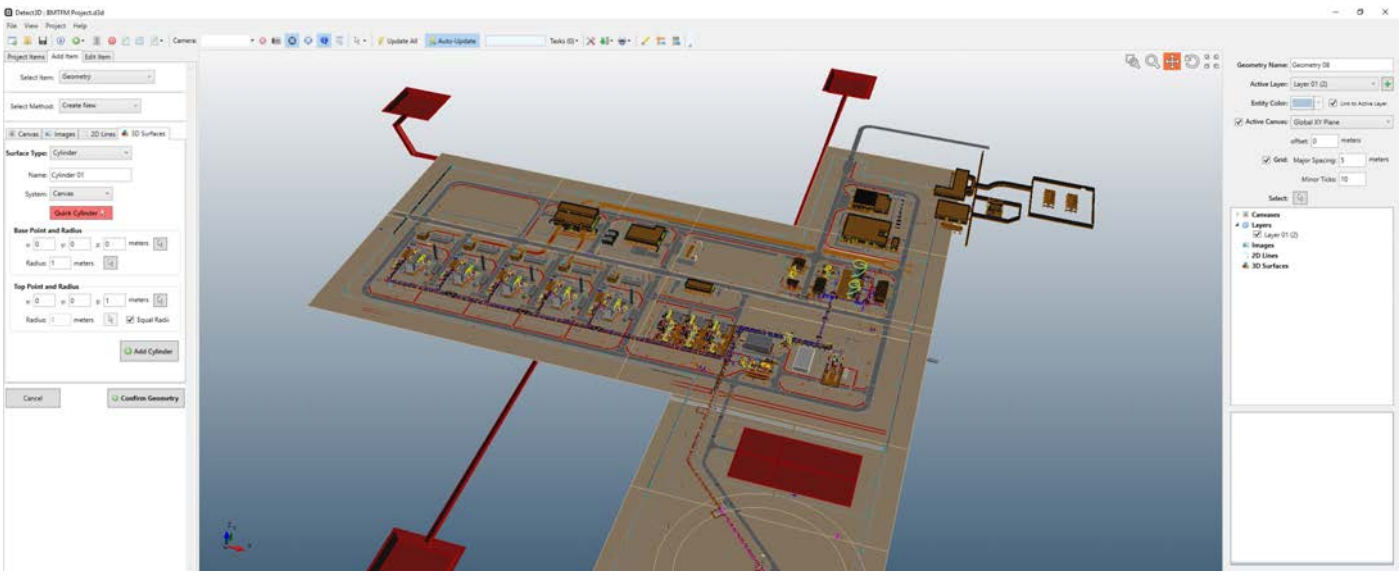


How to import plot plan layout drawings

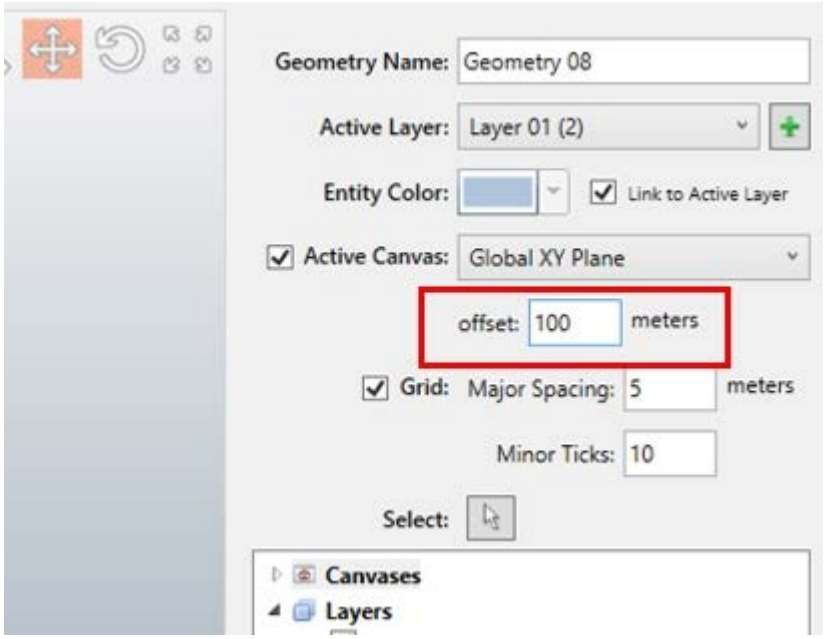
1. Convert the page from pdf to an image format (jpg or png) by using a screenshot tool.
2. Load your project, right-click on Geometry and select “Create New Geometry”



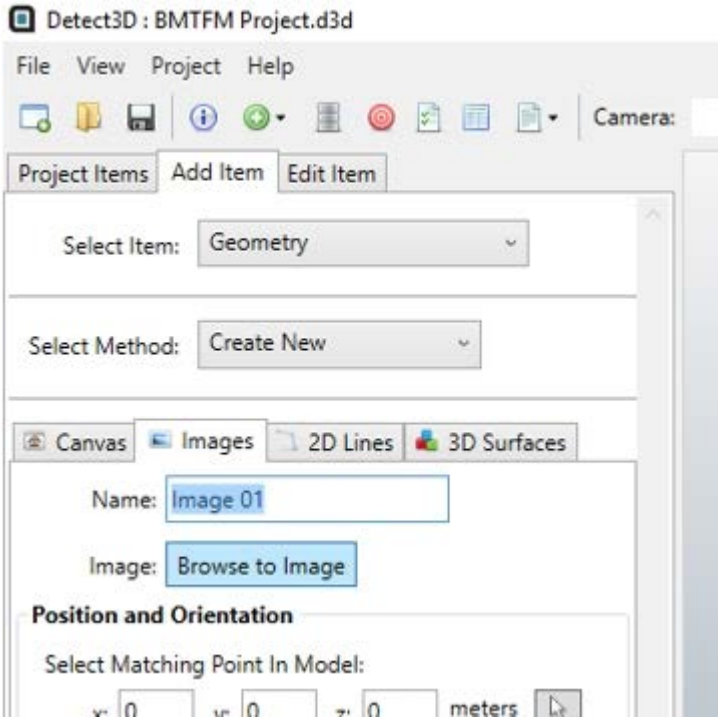
3. The CAD Creation Tool user interface, shown below, will appear. This is where you can add shapes, delete entities, and add images etc.



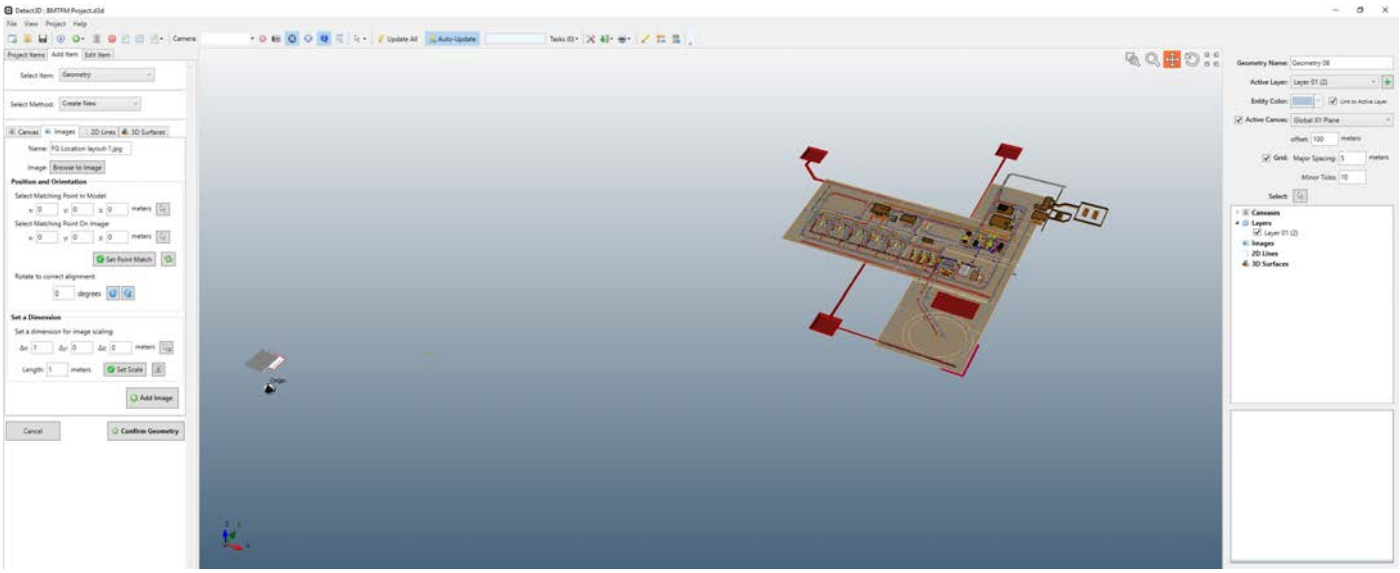
4. The first thing to get right is the “Drawing canvas”. This defines the plane that we are going to work on. For this example, the ground level for this CAD geometry is at about z = 100 meters, so select the offset (top right) to be 100 meters.

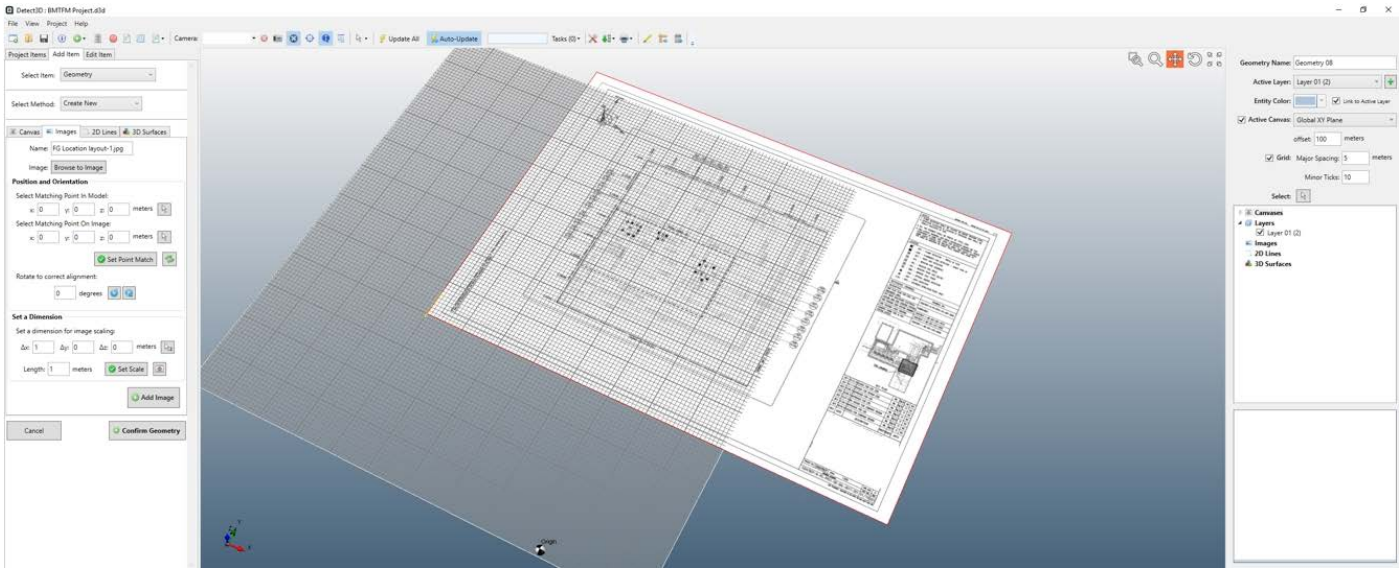


5. Next, on the left side select “Images”, click browse, and navigate to your saved image.



6. The image will be imported, but it will be a long way from the model – this is because the software doesn’t know the real location or the size of the contents of the image. If you zoom out, you will be able to see it.

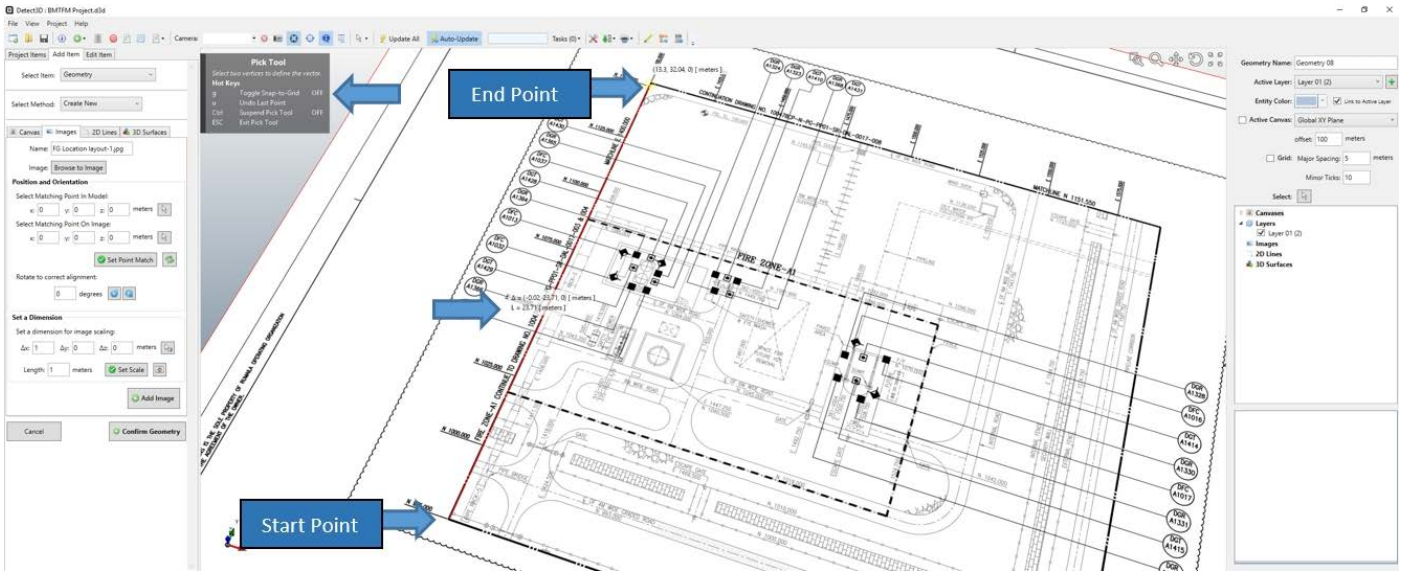




Zoomed in view of the imported file

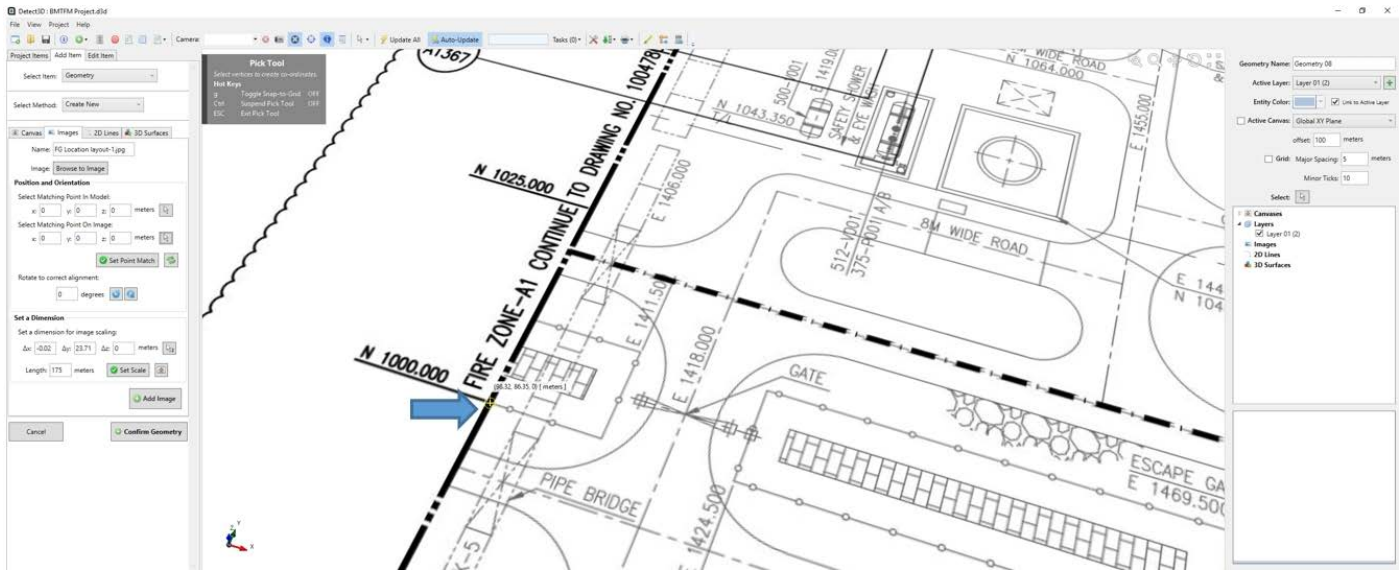
7. You can see the canvas and grid overlays on the image – turn these off to make things clearer – uncheck the "Active Canvas" and "Grid" checkboxes on the top right.
8. Now we are going to scale the image to the correct size. The trick is to find a good dimension on the image and use that to scale it. Fortunately, this drawing has the coordinates which exactly match the CAD file. We can use the North coordinates to scale the image. To do this, click the "Pick Tool" in the "Set a Dimension" group box on the Images tab. You will notice that the Pick Tool has a "2" on it – this means that it expects you to pick two points, i.e. the two ends of the dimension we're going to use for scaling:

TIP! Press "g" while the Pick Tool is active – this deactivates the "Snap To Grid" and allows you to pick the points more accurately.

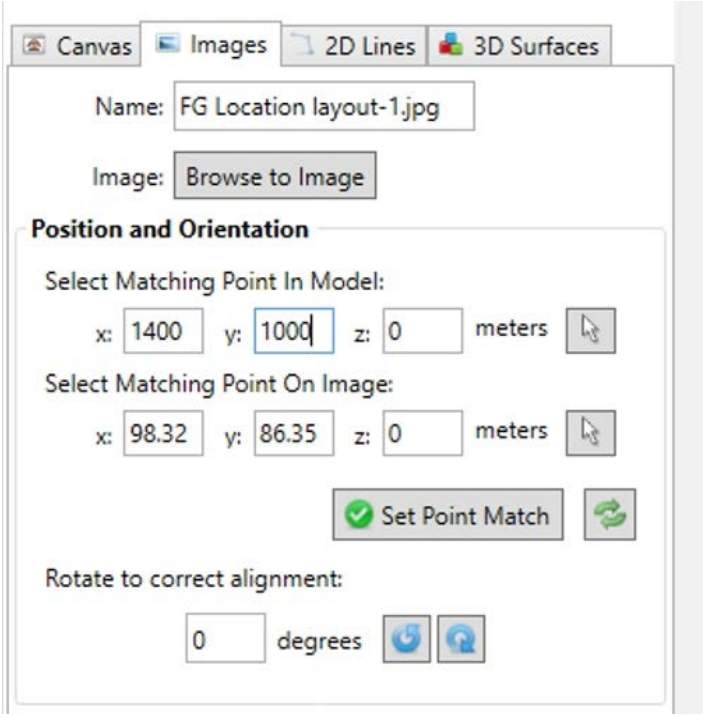


9. For this example, the size of the image is such that the selection above is 23.71 meters, but we know that the actual length is 1150 meters – 975 meters = 175 meters. So, overwrite the length to 175 meters and press “Set Scale”.

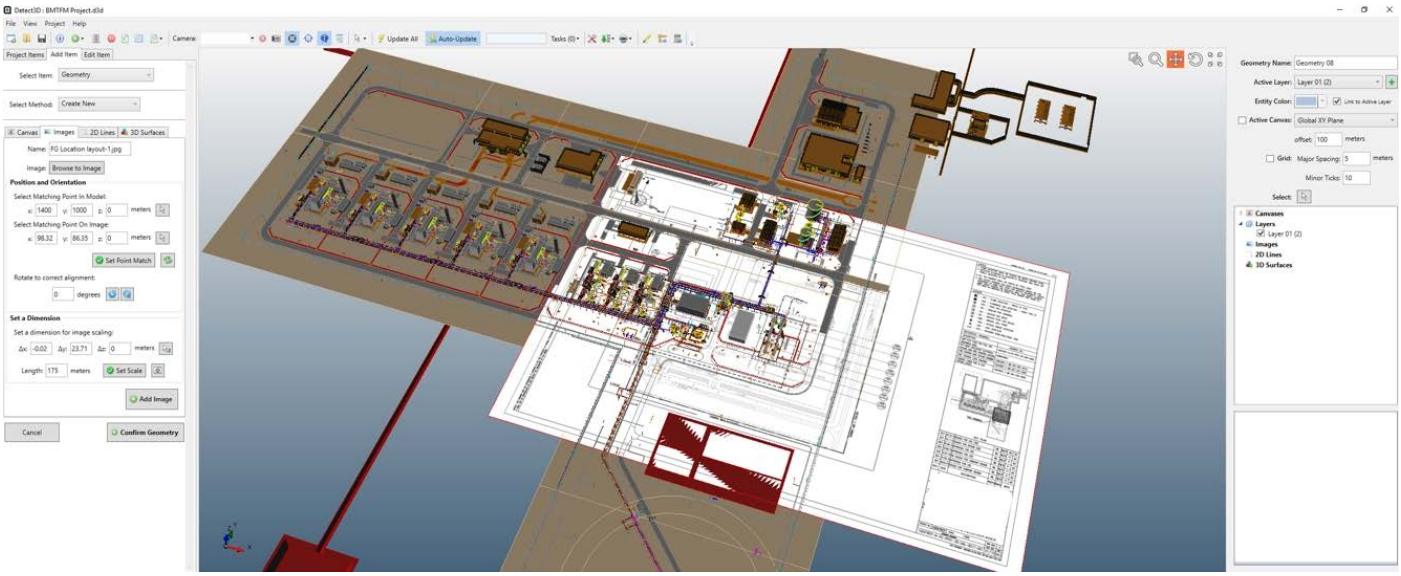
10. Zoom out to see that the image is now the right size! It’s still in the wrong place, so that need to be fixed. To do this, turn your attention to the “Position and Orientation” group box above the “Set Scale”. You will see two coordinate input boxes. The first asks you for a location on your CAD model, and the second asks you for the matching location on the image. For this example, the Pick Tool was used to pick out the N1000.00 E1400.00 point on the image:



11. With this point selected, type in these coordinates on the CAD file (x axis is East, y axis is North).

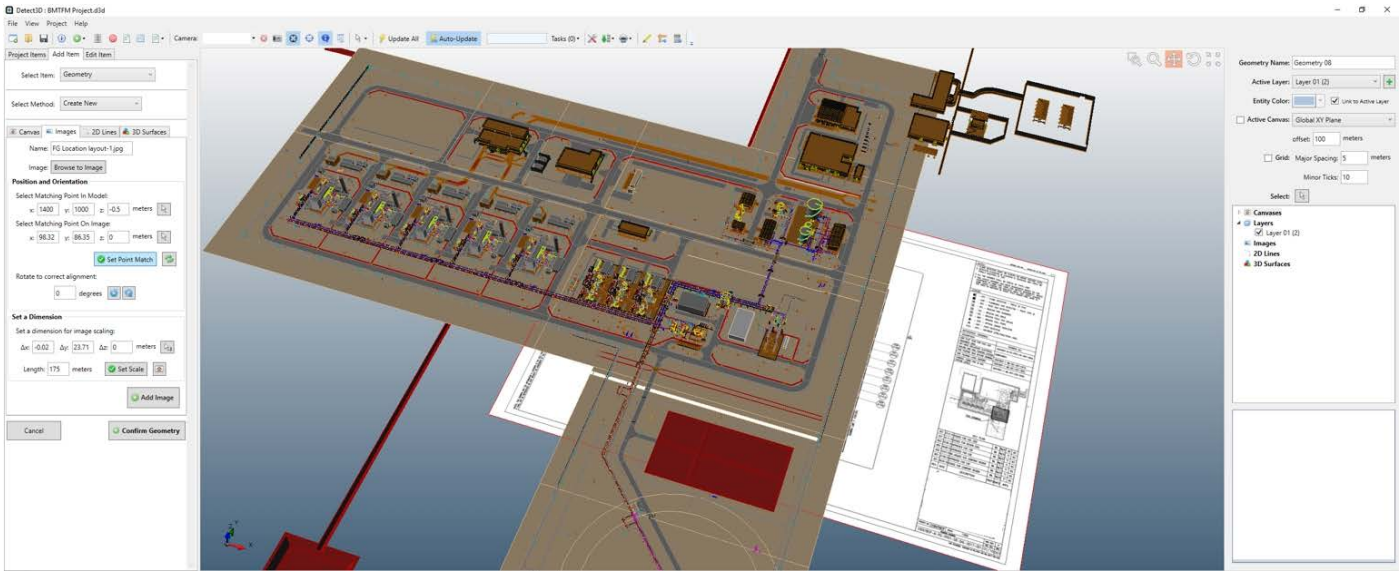


12. Once the points are typed in, click “Set Point Match” and the image will exactly align with the CAD file:

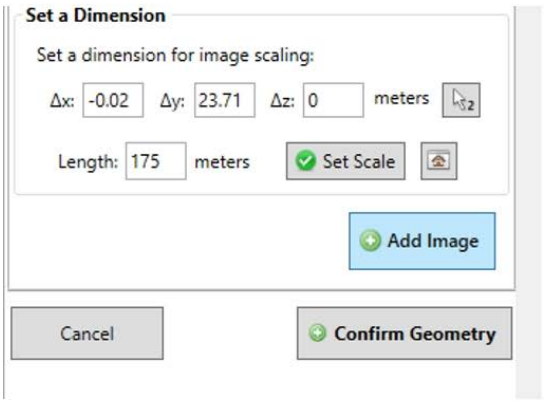


13. You can make slight adjustments to the z-coordinate of the point match to raise or lower the image to the desired height – sometimes when the image *exactly* lines on surfaces, the graphics card doesn’t know which to show and can lead to undesirable results.

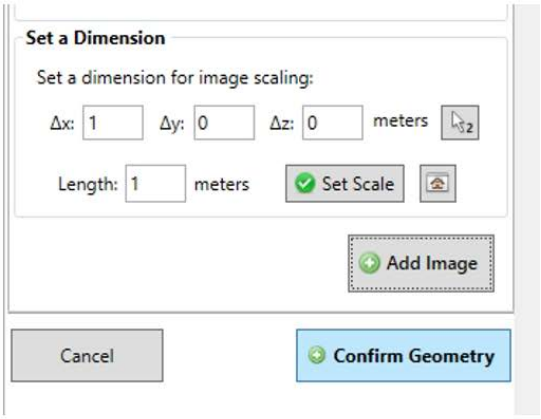
You can choose either to raise the image or lower it and make the ground levels transparent. To lower it, edit the “Matching Point in Model” to -0.5 meters (for example). The image should now be “below” the CAD:



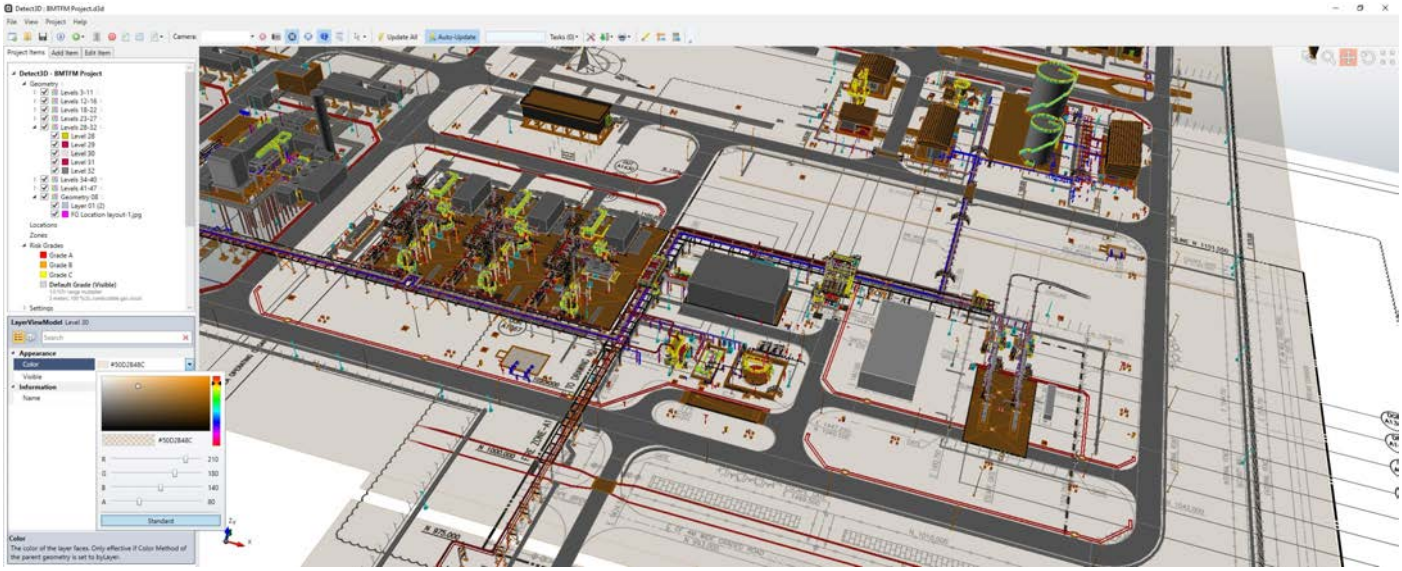
14. Click “Add Image” to add the image to the geometry you have created, then click “Confirm Geometry” to add the created geometry to the project. Don’t worry about the warning message, just click “OK”. You will notice that there is a new geometry listed in the project tree.



…then…



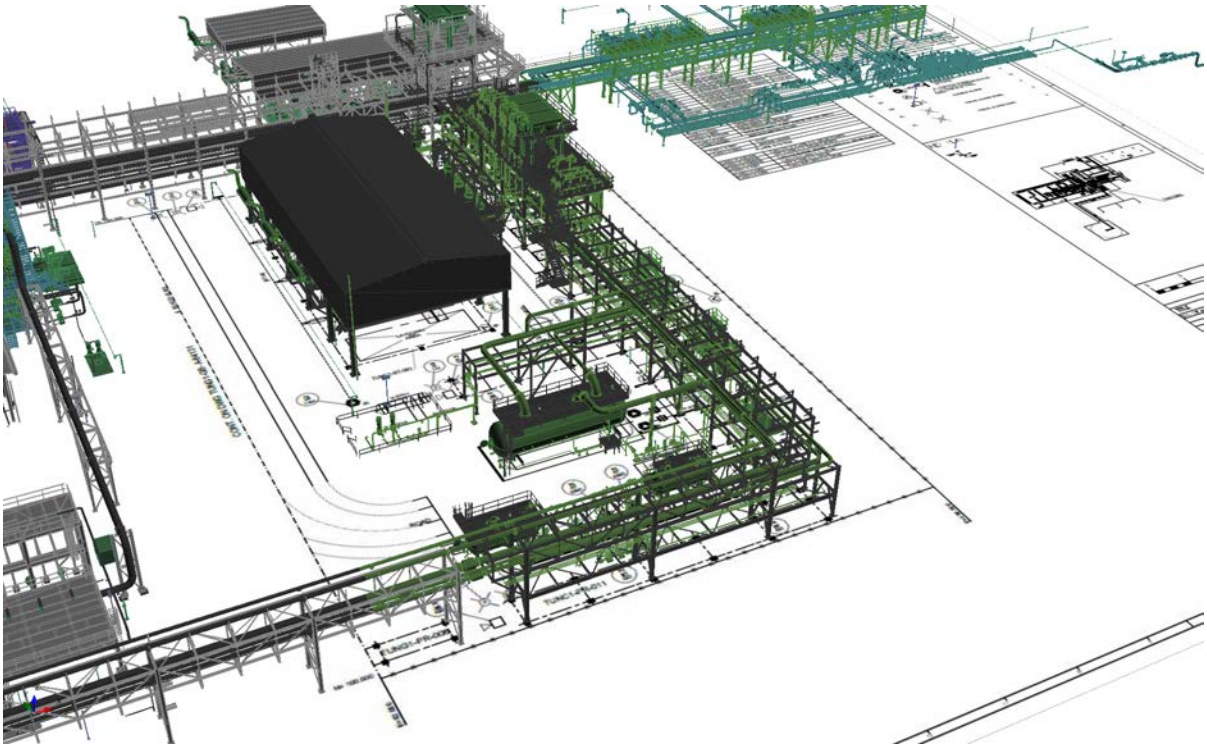
15. You can then play with the geometry and transparencies via the 'alpha' channel of the layer property to get the desired effect. If the image being lower than grade doesn’t work, you can always add the image again above grade etc. In the image below, the transparency was changed for one of the layers of the CAD



This can be done with more layers until you are pleased with the result.

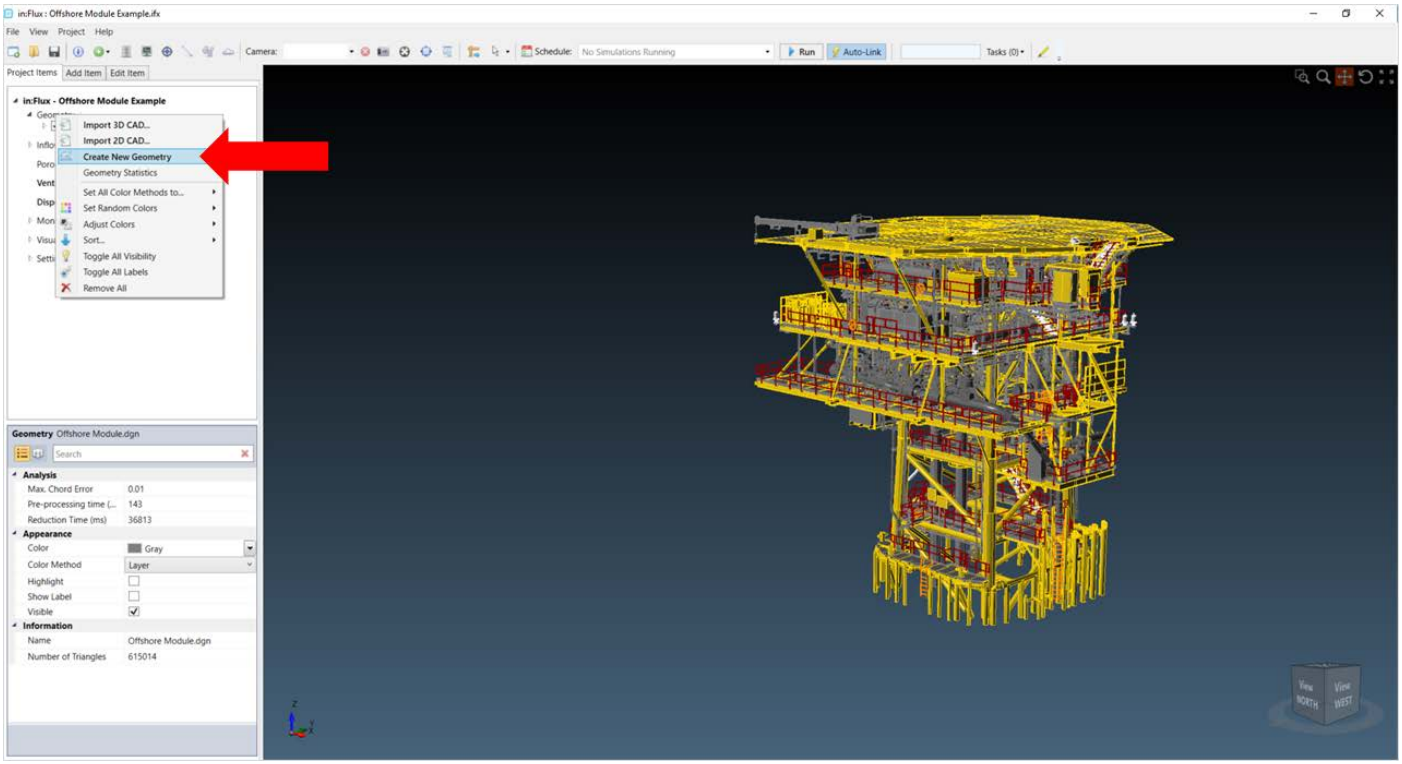
Importing Images as Ground Objects

Google Earth screenshots or images of the environmental surrounding of a facility can be added to Detect3D or in:Flux projects for visualization purposes. Users may also choose to add plot plan drawings (example below) rather than environmental images.



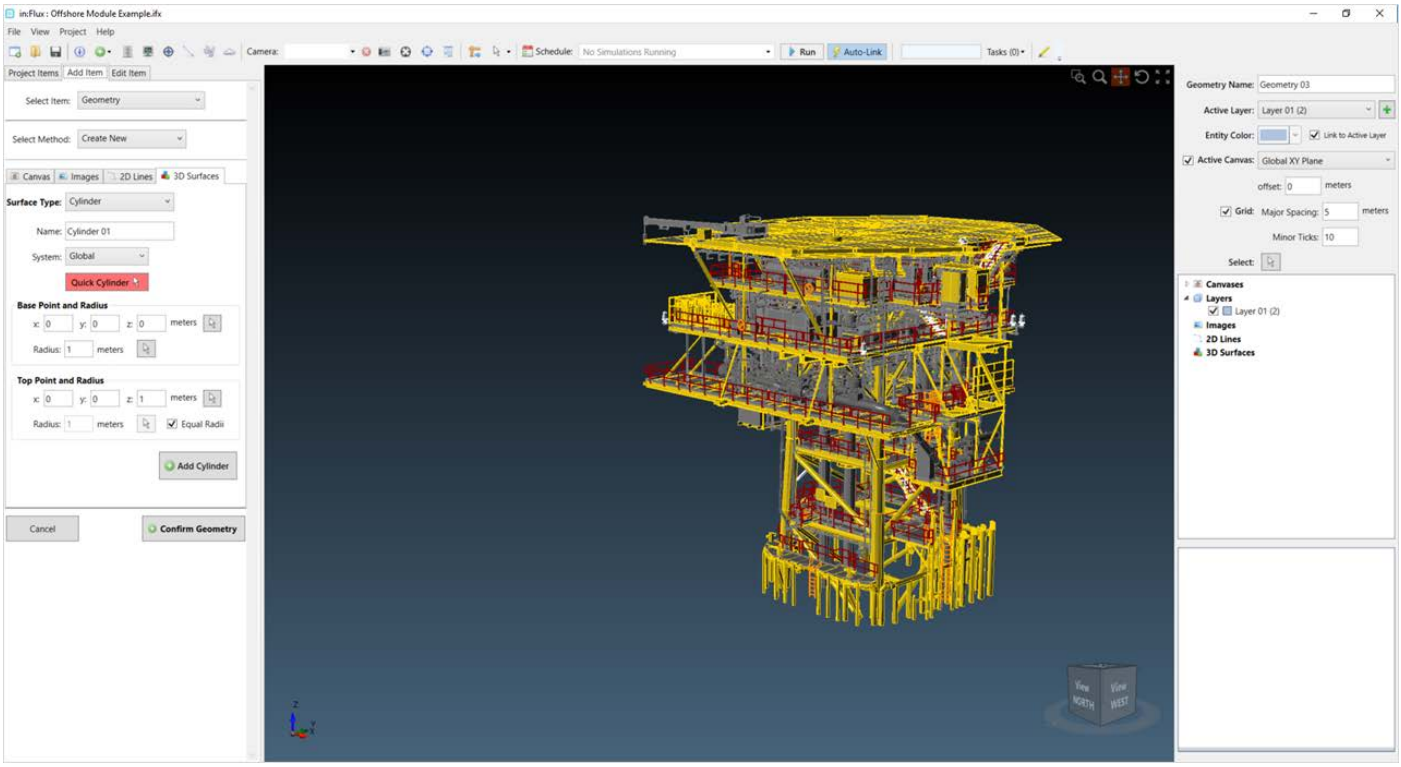
The steps below will go over adding an HD image of the ocean surface to an offshore module.

1. Create and save an image or screenshot into jpg or png format
2. Load your project and right-click on Geometry and select “Create New Geometry”



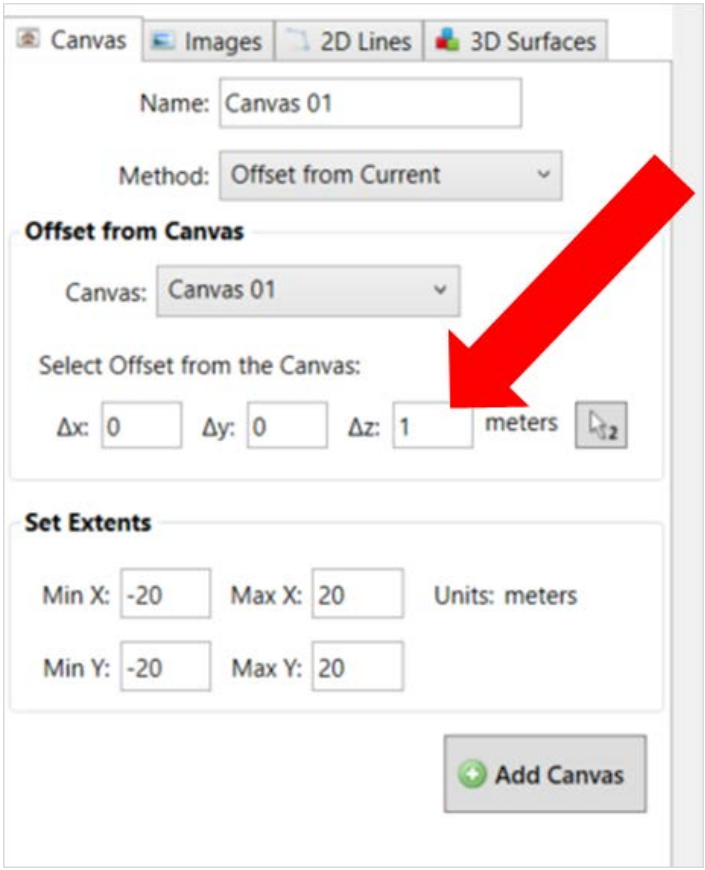
Tutorial 6 - Figure 55 - Selection of 'Create New Geometry' after right-clicking on the Geometry header

3. You will see the user interface for the geometry creation capabilities. This is where you can add shapes, delete entities, and add images etc.



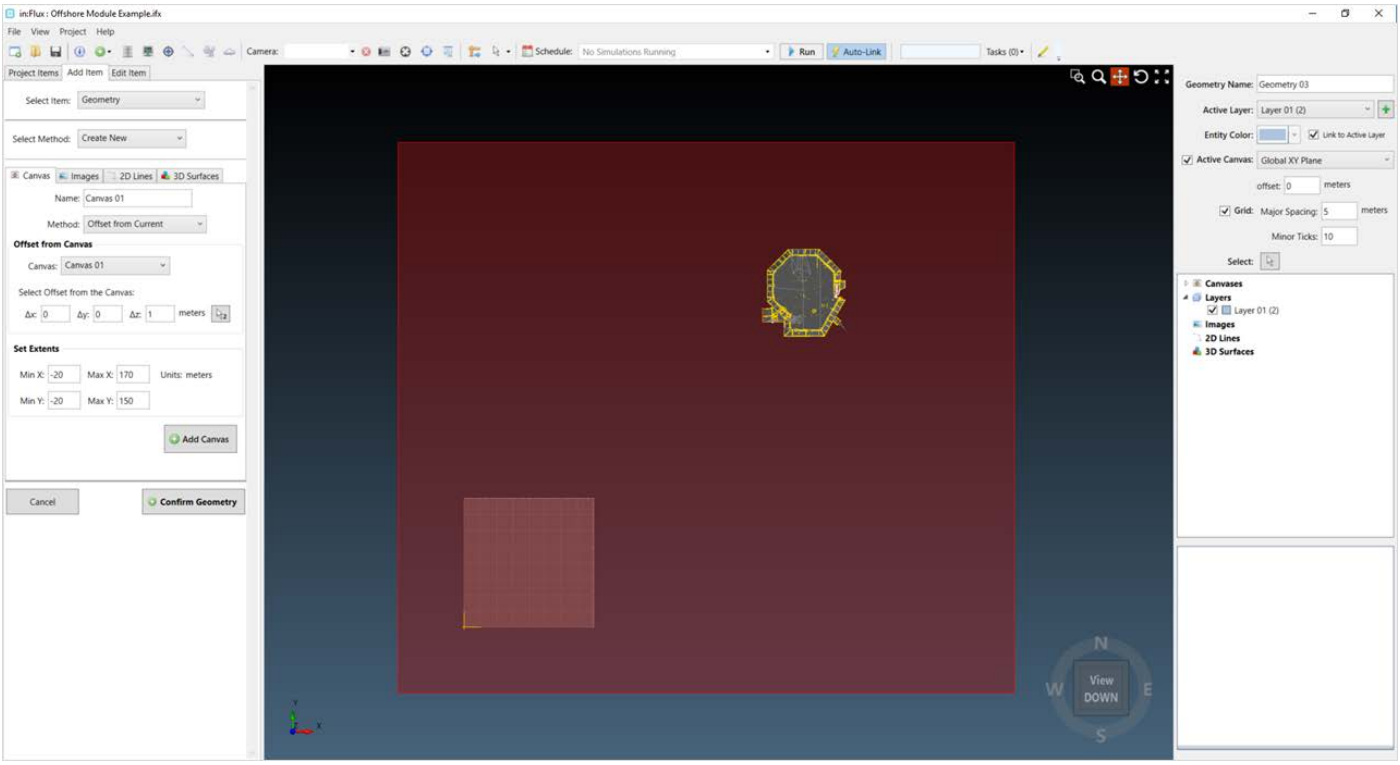
Tutorial 6 - Figure 56 - Create CAD interface

4. The first thing to setup is the “Drawing canvas”. This defines the plane to work on.
    a. Select the **Canvas** tab on the left-side of the screen.
    b. Enter a name for the Canvas or just leave it as "Canvas 01"
    c. Specify the **delta-z** value to be the height of the plane which you want the image to be imported onto. This geometry has already been positioned so that the base of the CAD file is at a height of z = 0 meters which is the "ground" or ocean surface for this geometry.



Tutorial 6 - Figure 57 - Setting the height of the Canvas on which the image will be imported.

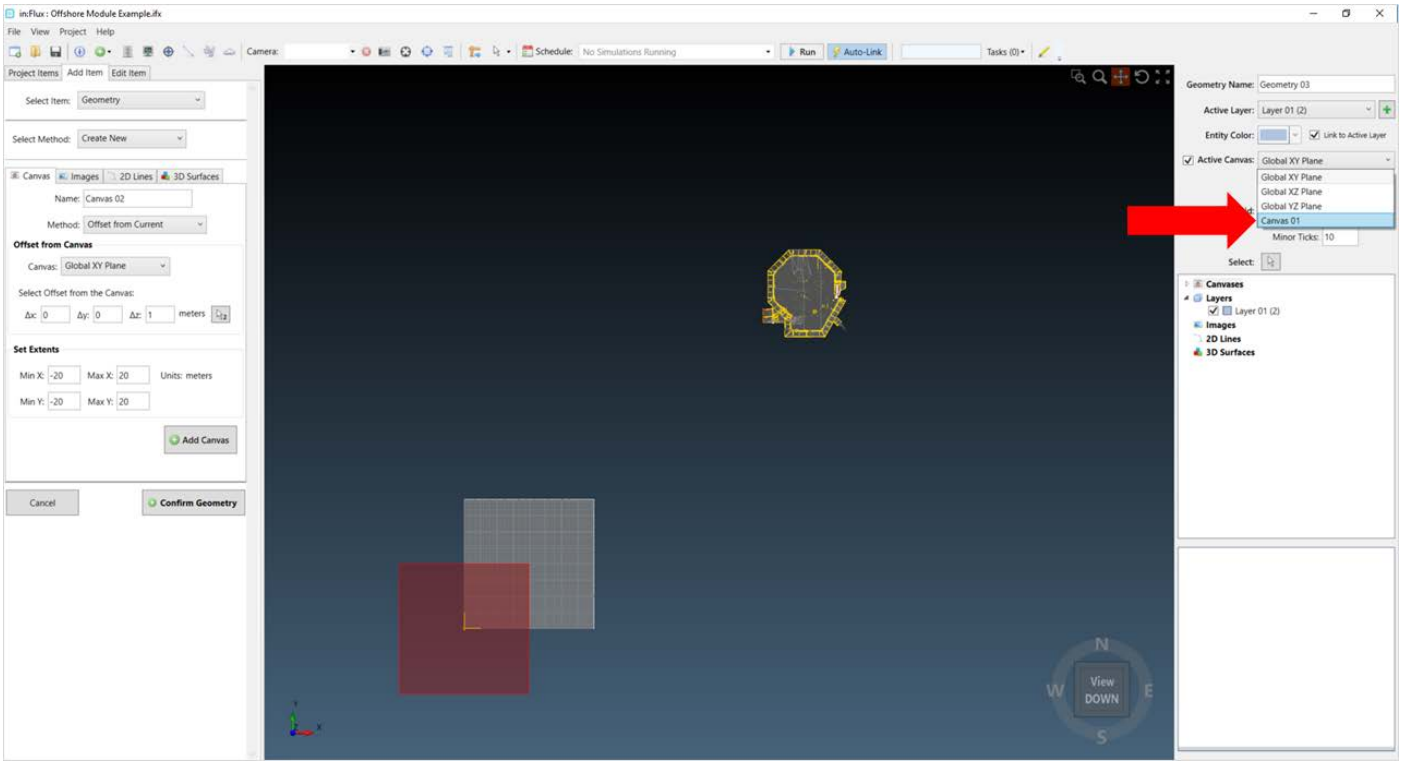
a. Right-Click in the 3D window and select the **View From** option as *Top View.* Change the **Min X, Max X, Min Y and Max Y** values to encompass the CAD file.



Tutorial 6 - Figure 58 - Setting the canvas to encompass the CAD model

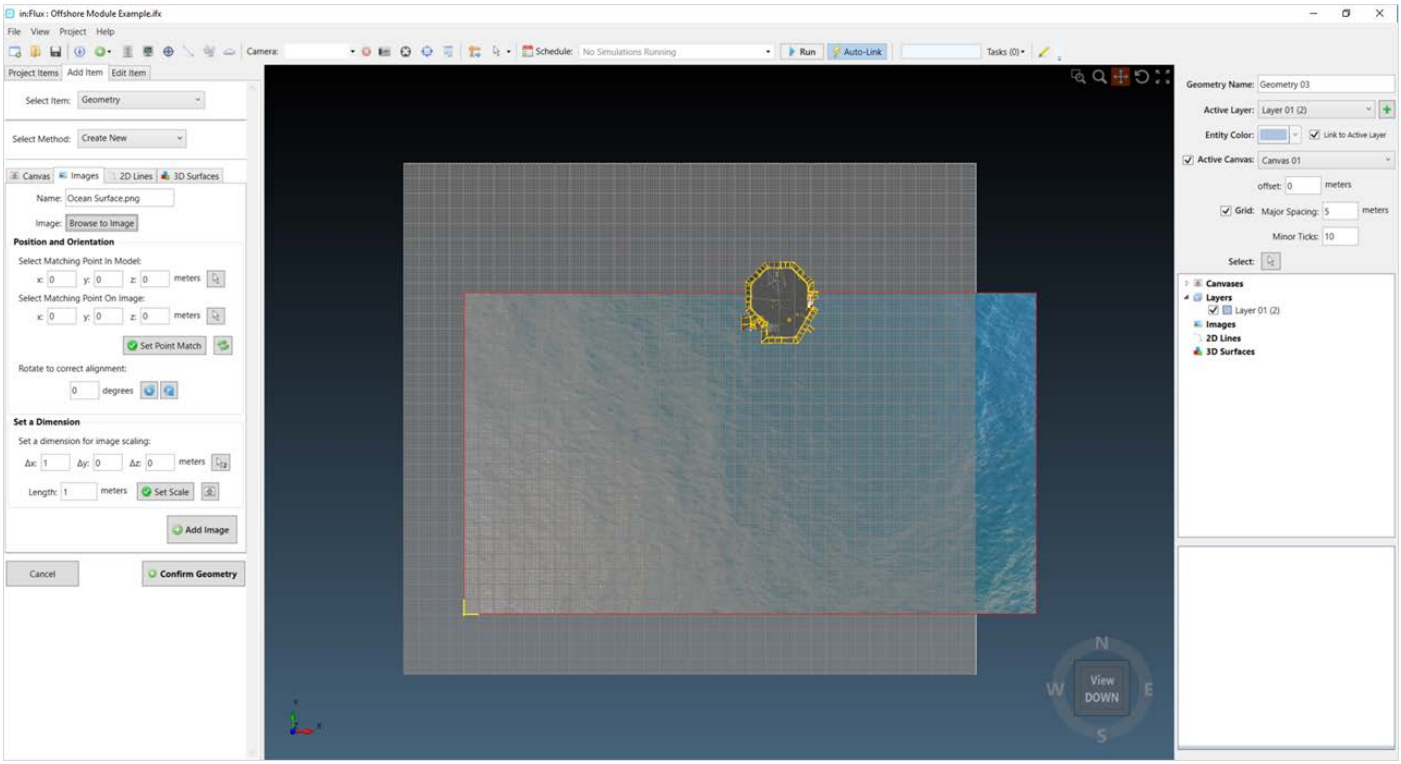
a. Click **Add Canvas**

5. Now that the canvas has been defined, we need to make it "Active" so we can draw upon it. In the upper-right side of the screen, choose your named canvas from the **Active Canvas** dropdown menu.



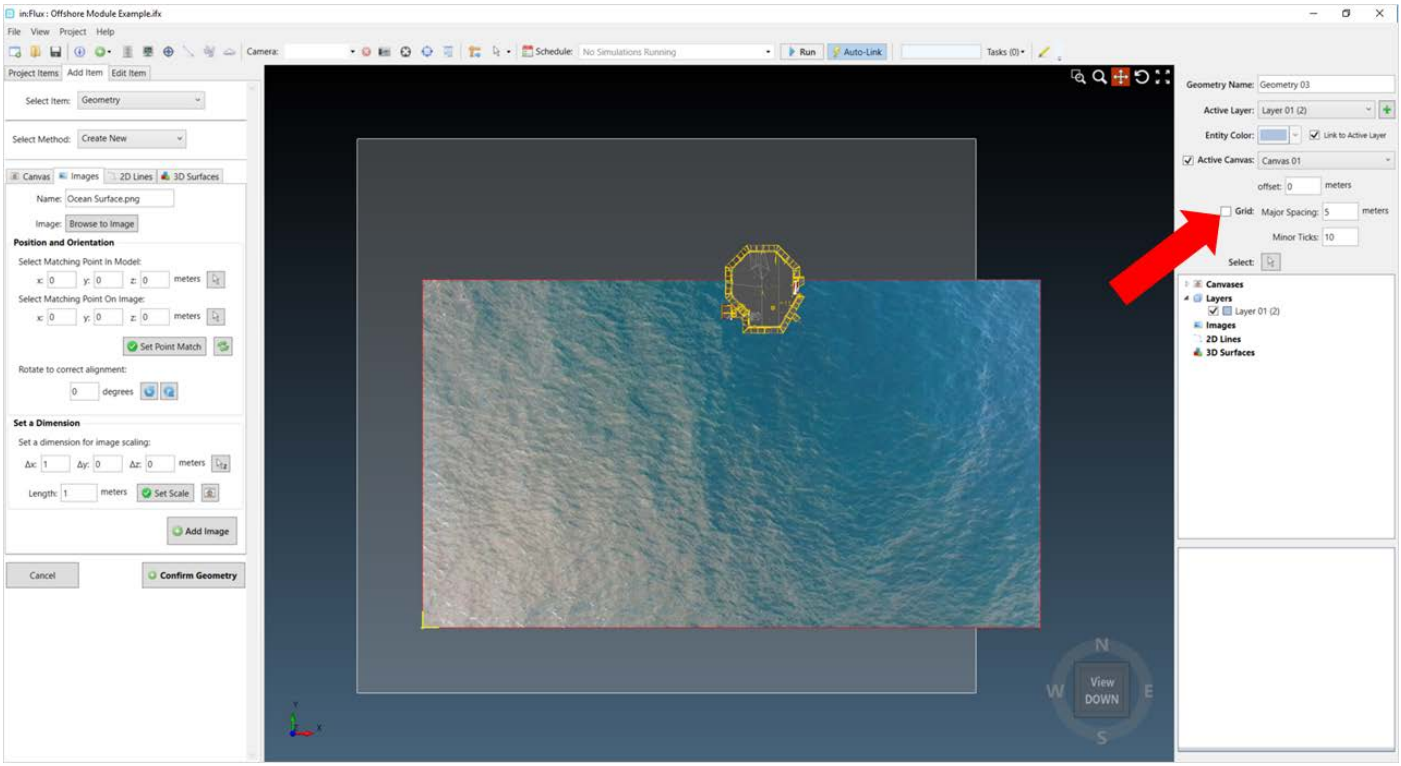
*Tutorial 6 - Figure 59 - Selection of the newly defined canvas from the **Active Canvas** dropdown menu*

6. Next, on the left side select the **Images** tab, click **Browse to Image**, and navigate to your saved image.
7. The image will be imported, but it will be positioned at the origin and scaled based on estimates from the active canvas – this is because the software does not know the real location or the size of the contents of the image.



Tutorial 6 - Figure 60 - Ocean Surface Image has been added to the project but is not positioned in the desired location

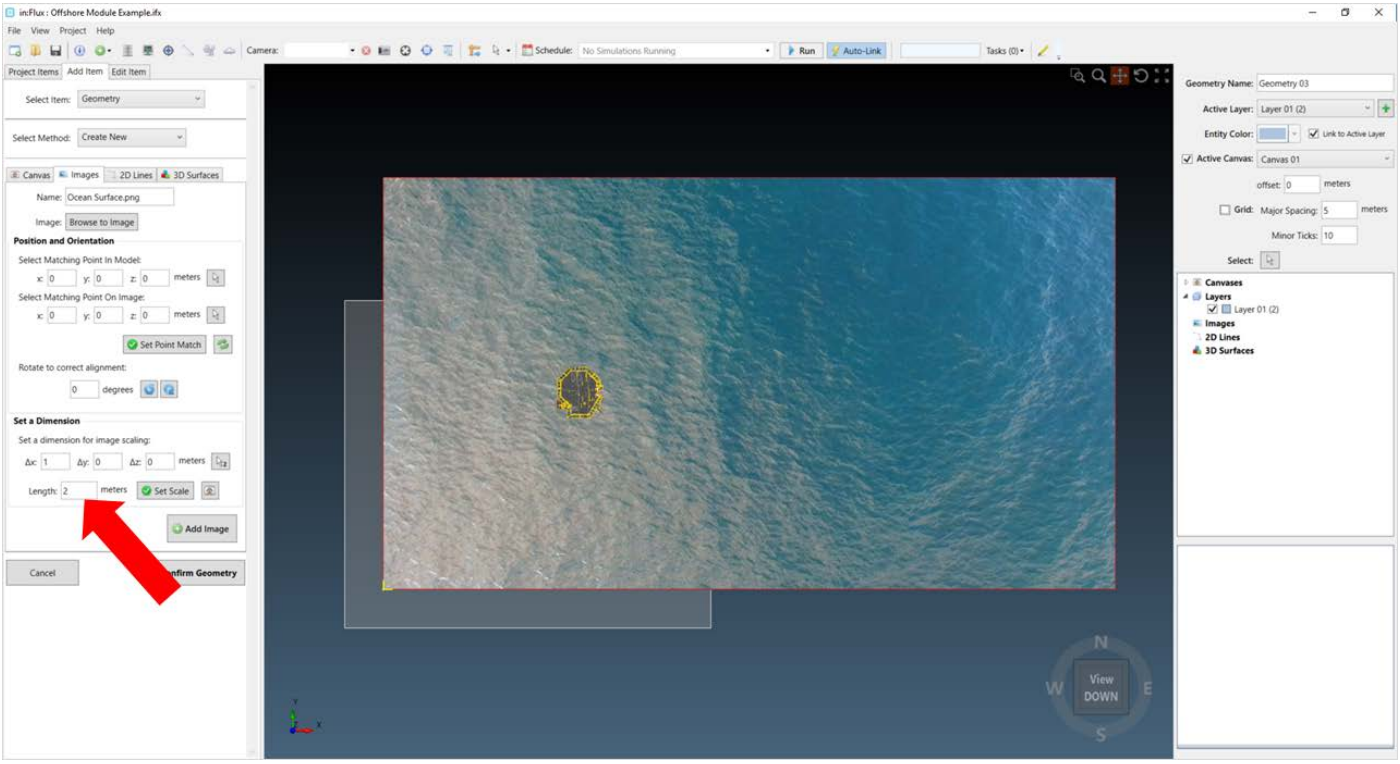
8. You can see the canvas and grid overlays on the image – turn these off to make things clearer – uncheck the **Grid** checkboxes at the upper right of the window.



Tutorial 6 - Figure 61 - Toggling off the grid to make the imported image easier to see

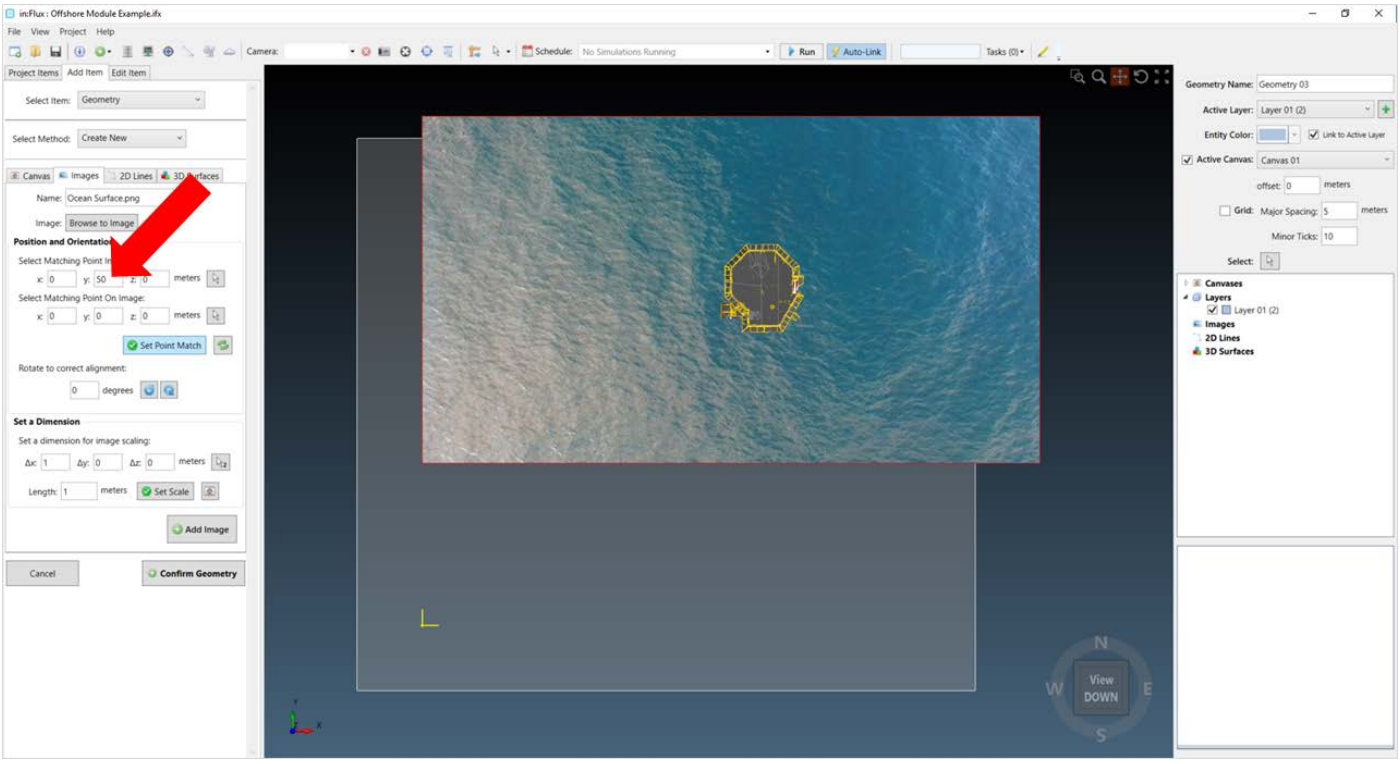
9. Now the image needs to be scaled to the desired size. Since we are only importing this image for aesthetics, we want to make sure the image encompasses our geometry. However, you may want to import a plot plan drawing instead of a picture which requires the scale and positioning to be correct. For this image, the scale seems reasonable, but it is not encompassing the geometry. We can either scale the image to be larger or move the image in the positive y-direction. Note: each image import will be different depending on the origin of the imported geometry.

    a. To scale this image, a value of 2 was entered in the **Length** text box in the **Set a Dimension** section. This doubled the size of the image as the delta vector was (1, 0, 0) with an original length of 1.



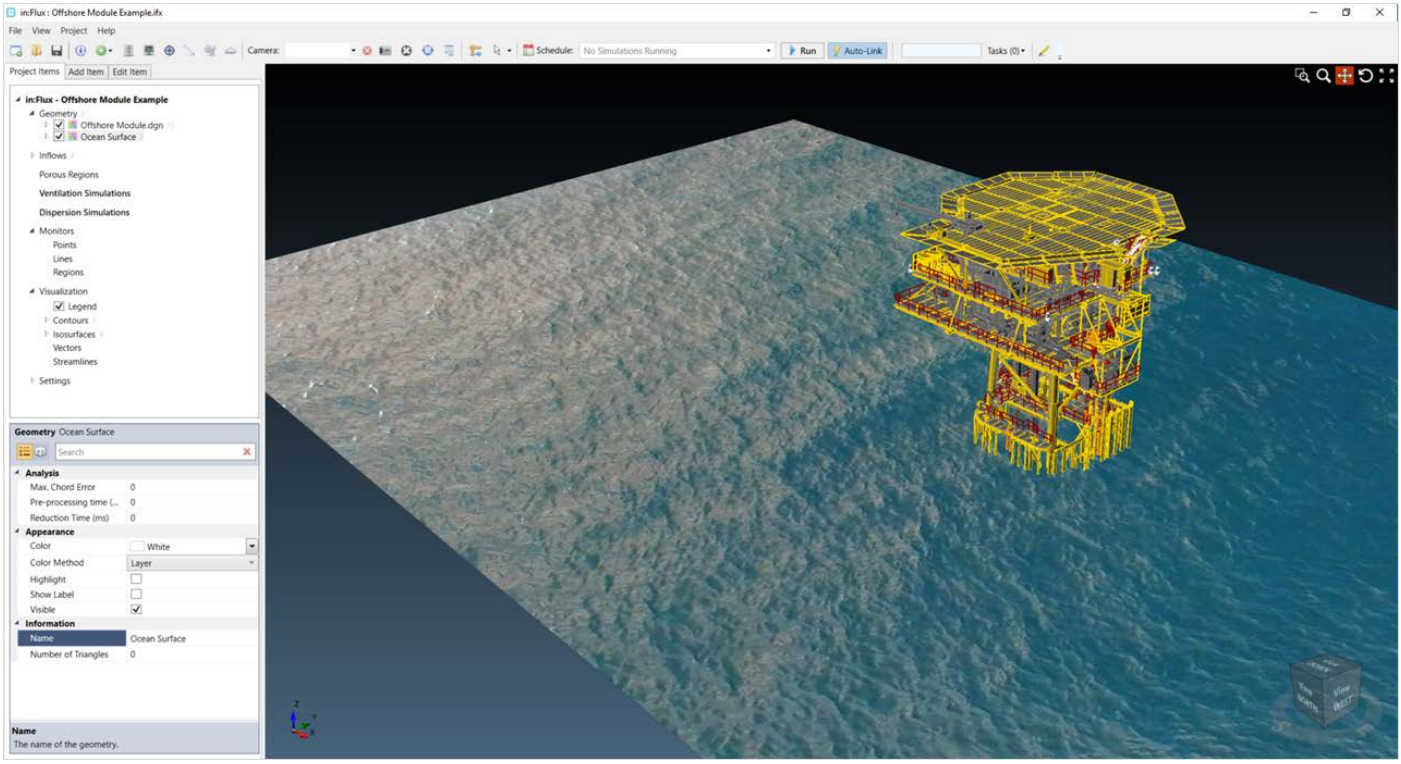
Tutorial 6 - Figure 62 - Indication of how to scale an imported image

a. Alternatively, to move the image in the positive y-direction, change the Y-value of the **Select Matching Point in Model** on the left side of the window to be the distance you want the image moved. For this example, the Measure Tool was used to determine that the image should be moved 50 meters. Once the value is entered click **Set Point Match** to translate the image.



Tutorial 6 - Figure 63 - Showing where to translate the image in the positive y-direction

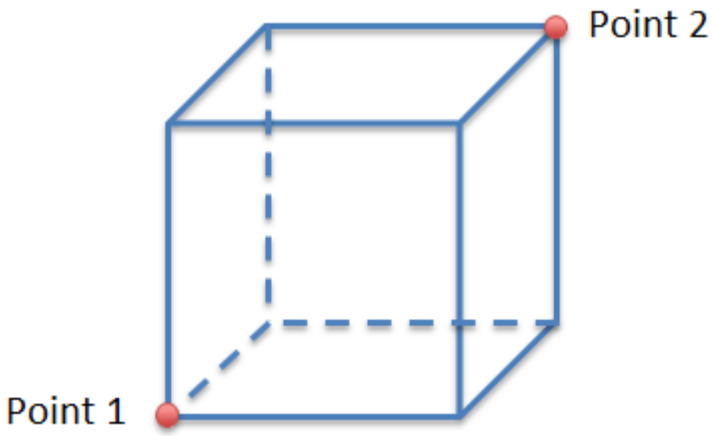
10. Regardless of the positioning method, once the image is in the desired location and with the desired scaling, click the **Add Image** button and then **Confirm Geometry.** Click **OK** on the warning message. You will notice that there is a new geometry listed in the project tree containing the imported image. You can toggle this on and off as with the other project items. As we did not add any CAD entities in this section, the image will not affect any calculations. You can rename the geometry from its properties panel.



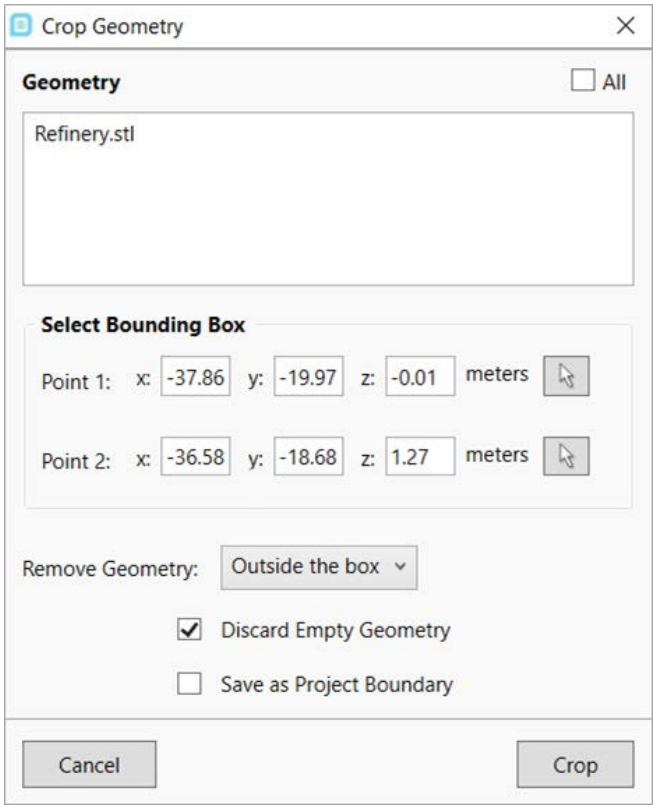
Tutorial 6 - Figure 64 - Showing the imported image of the ocean surface in the project

CAD Geometry Crop Tool

For managing projects with large CAD files, Detect3D and in:Flux comes included with a crop tool capability. The tool allows you to define a three-dimensional box, based on two points (shown below), and either keep the geometry inside or outside the box.



You have the option to crop a single geometry file or multiple geometry files at one time by selecting the *All* checkbox at the top right of the Crop Geometry window. Selecting the *Save as Project Boundary* option will save the dimensions of the cropping box so when you load a new CAD file it will be cropped to the set Project Boundary.

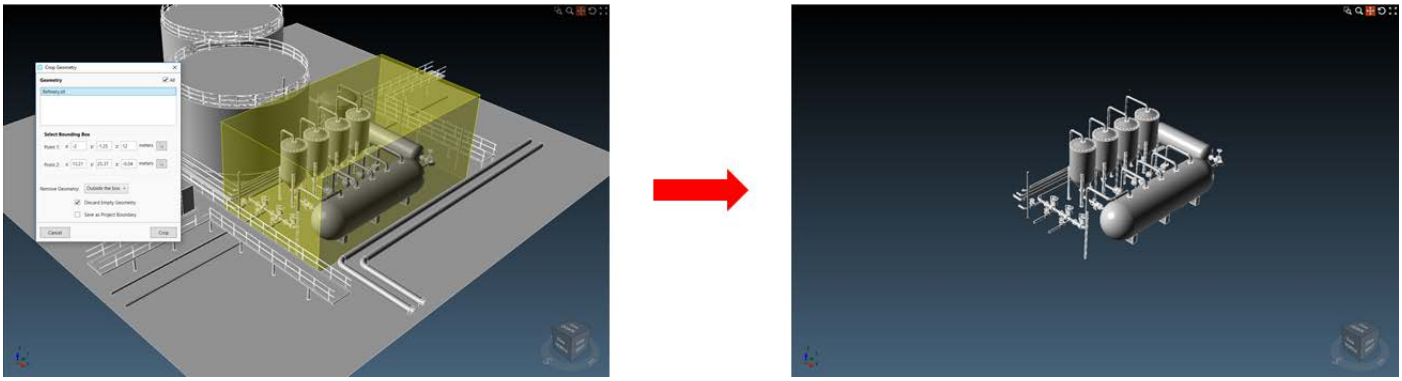


The Crop Tool can be accessed by clicking *Project* from the menu bar and selecting **Crop Geometry**. A window (shown above) will appear which allows you to use the Point Pick Tool to select two opposing points of the defined cropping box.

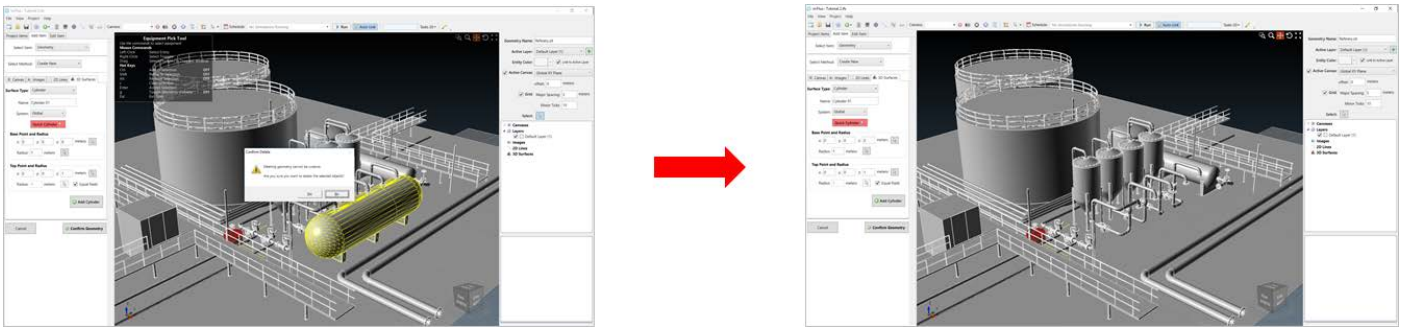
Removing CAD Pieces in Existing CAD Files

Occasionally geometry files have unwanted pieces e.g. human models, text names, superfluous equipment, etc. Both Detect3D and in:Flux comes with built in capabilities to remove individual CAD pieces as well as entire sections of CAD geometries.

Using the Crop Tool, you may define a cuboid region to remove CAD outside or inside the defined region.

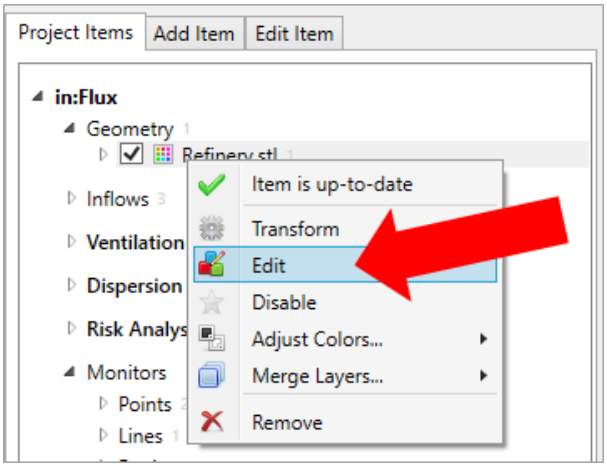


Alternatively, you may use the Equipment Pick Tool (described below) in the *Create New Geometry* interface to define the CAD pieces (triangles) that you desire to be removed and press **Delete** on your keyboard. You will be prompted to confirm that you want to remove CAD pieces before the deletion is made. It is recommended that you save your project prior.

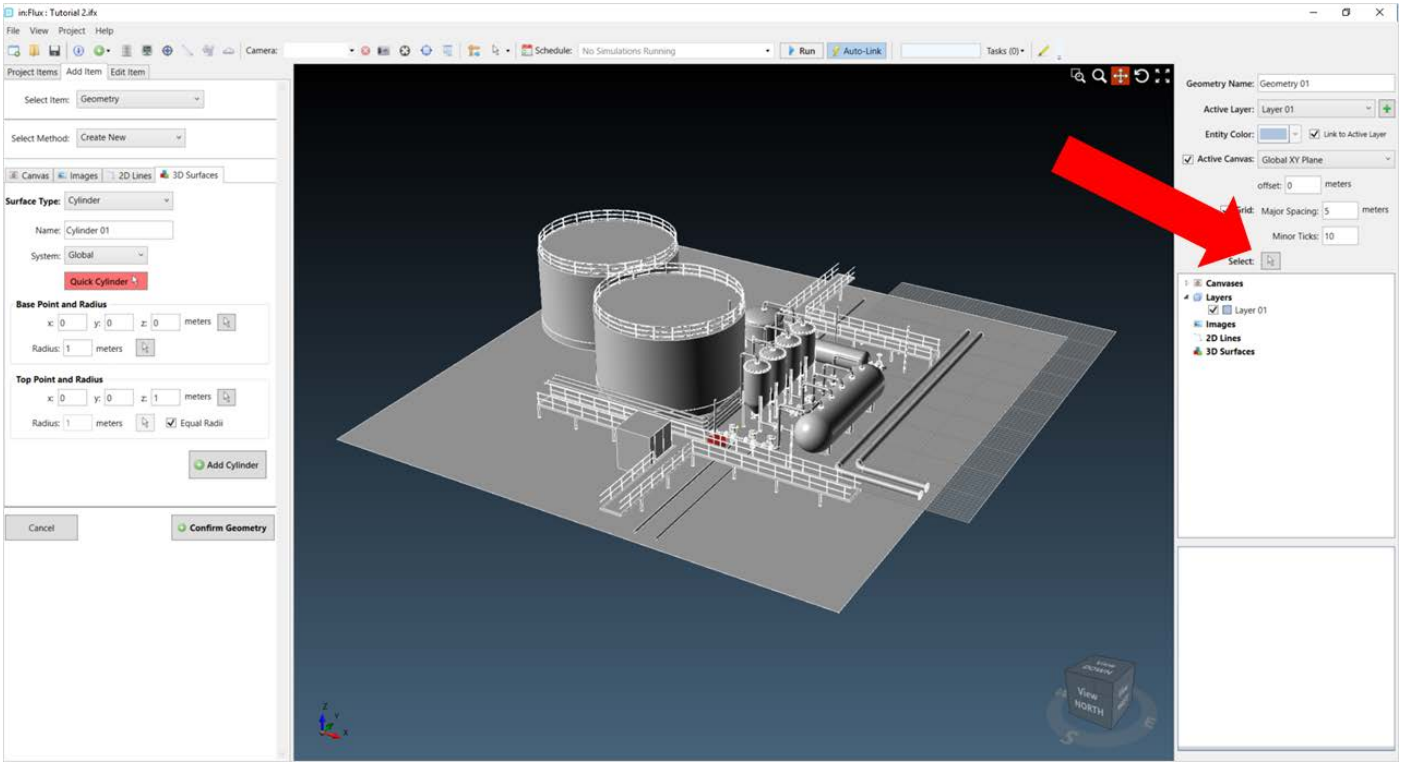


Detailed Description of Removing Specific CAD pieces from your project:

Right-click on the geometry that you want to remove entities from and select “Edit”.

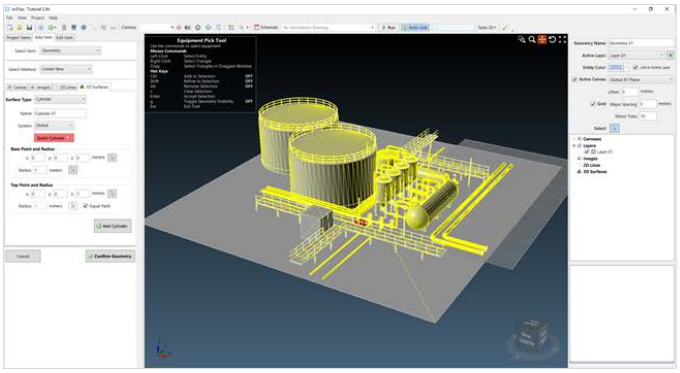


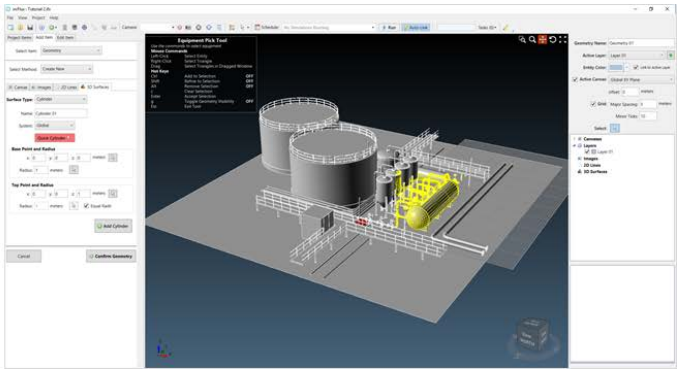
You will see a new user interface, which is used for adding and editing geometry. The Equipment Pick Tool in can be used when removing CAD pieces from an imported geometry file. The symbol  is the same as the pick tool icon and is located in the Geometry Creation Tool, shown below.



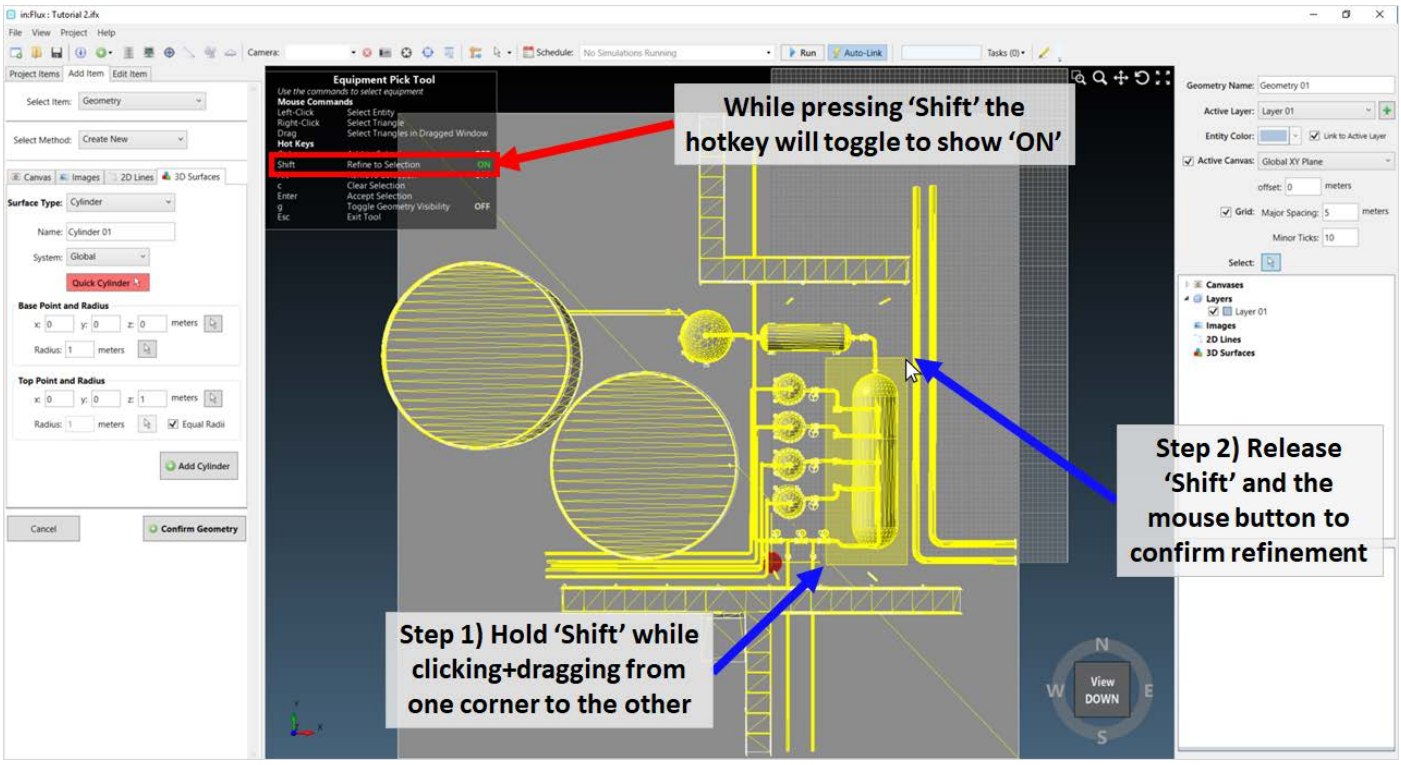
Once activated, the upper left corner of the viewport window will display a hotkeys list (shown below) which aide in selecting pieces of equipment.

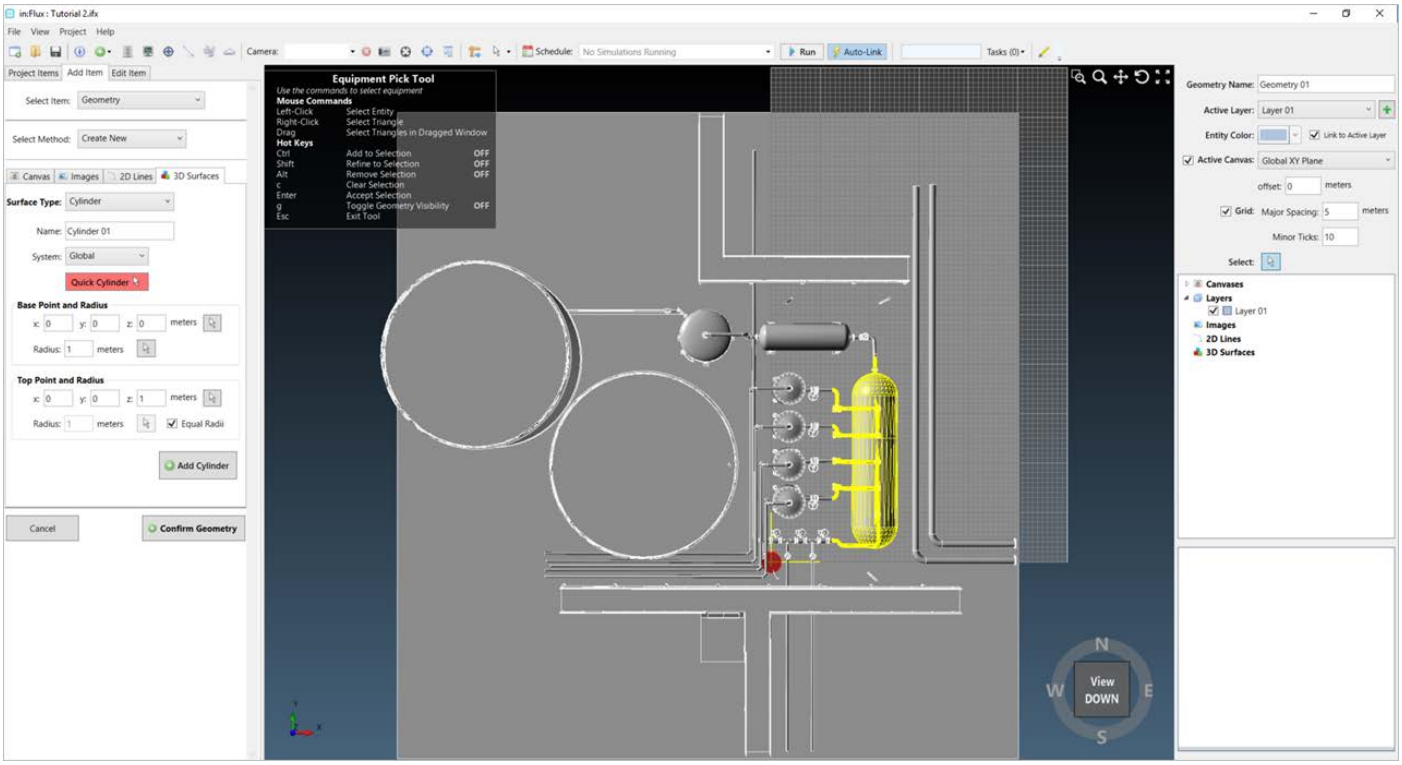
Pressing and holding the **Ctrl** key and clicking once on a layer or entity in which you want to select is the best way to begin selecting a piece of equipment. If there is only one layer in the CAD file then the entire file will be selected. In the example below the **Ctrl** key was pressed and the left mouse button was used to click the large tanks in the geometry. After clicking, the software determines all the triangles associated with that layer or entity. You may also press **Ctrl** and **click+drag** over a region you want selected. The two images below show the resulting selection from just clicking (left) and click+drag (right). If you can't see the selection, press "g" on the keyboard to toggle the visibility of the geometry.



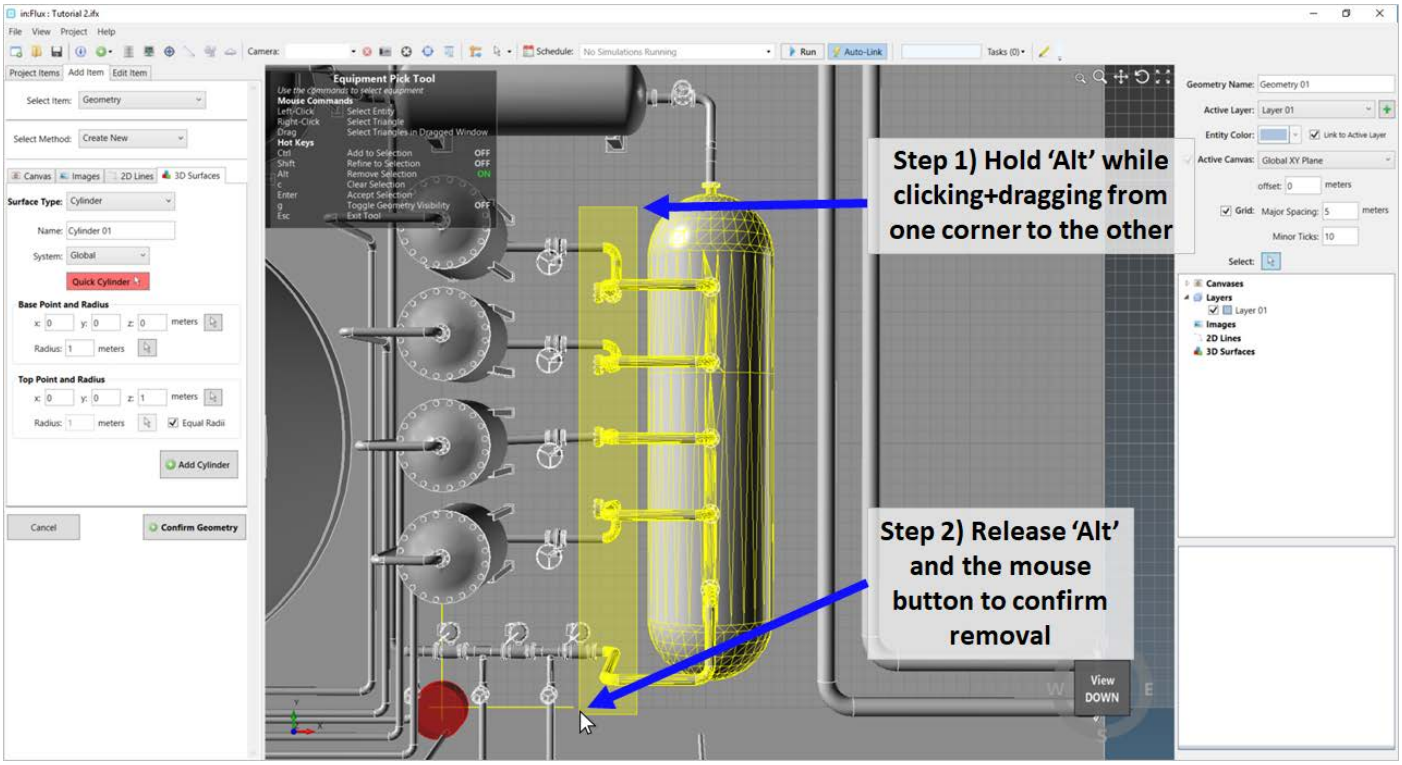


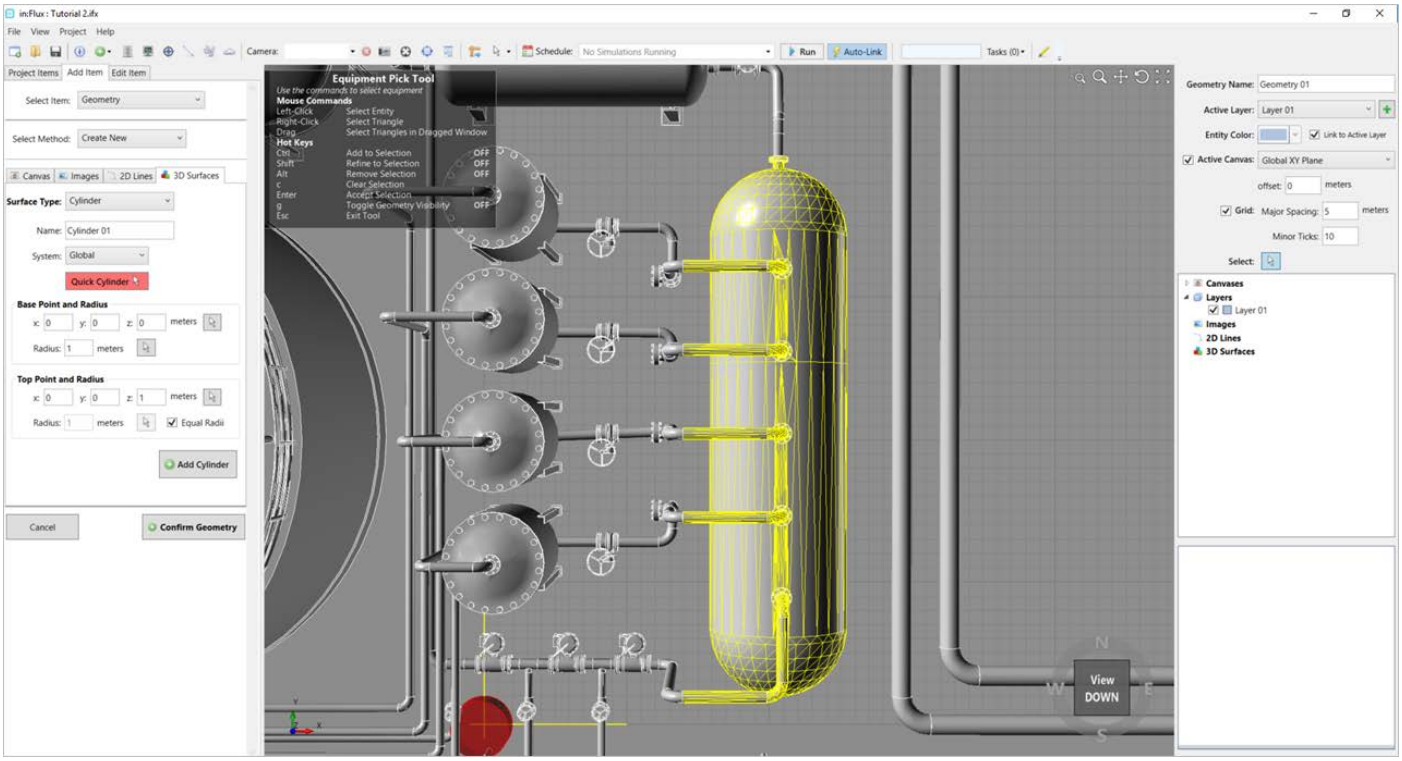
At this point you can choose to use the **Shift + Drag** to refine the existing selection. Everything contained in the dragged box will remain in the selection. Using this paired with different views of the 3D window significantly helps in dragging over regions of importance. Below a Top View has been used to assist in drawing the refinement region.



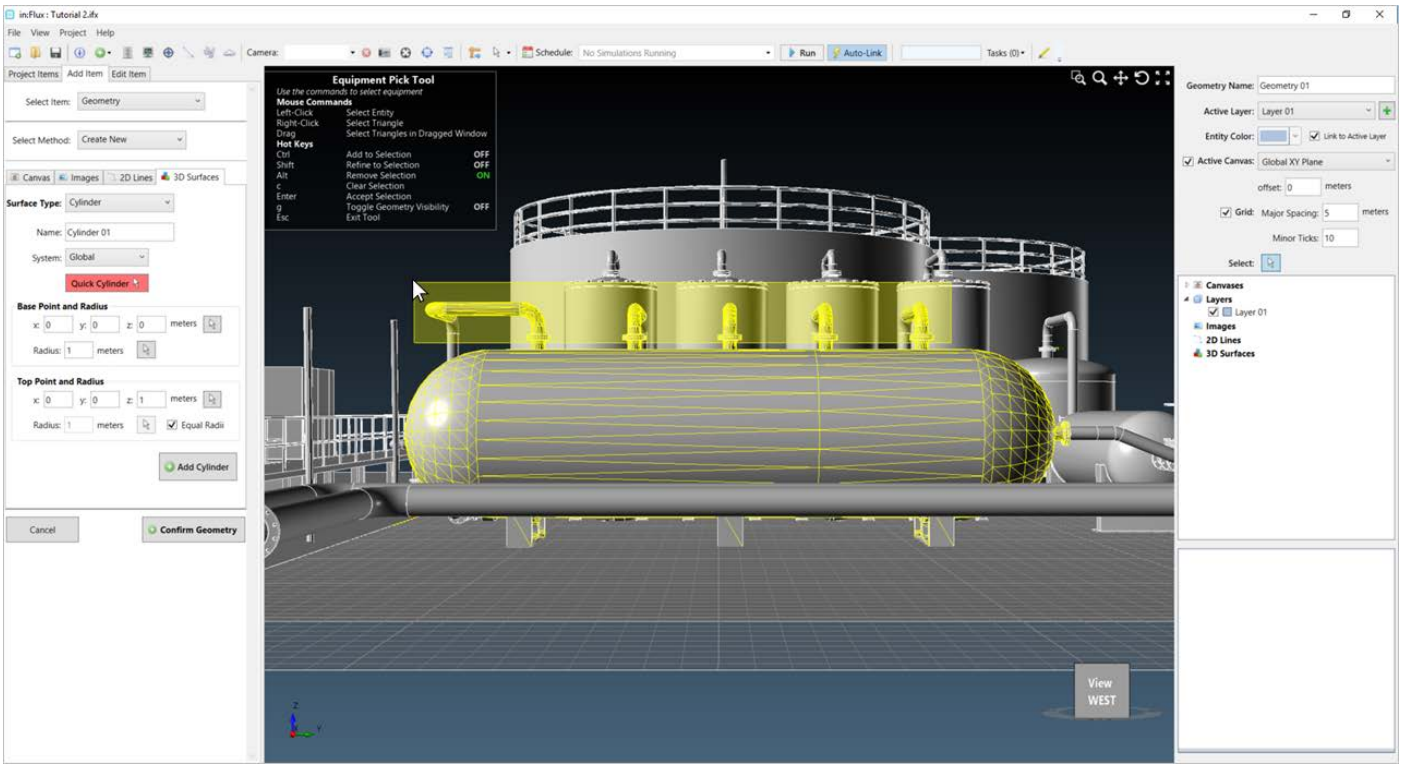


Alt + Drag will remove the dragged selection from the existing selection. As an example, in the below image, the **Alt** key was pressed while drawing a box over the flanges on the left side of the tank. This will remove the flanges from the selection.

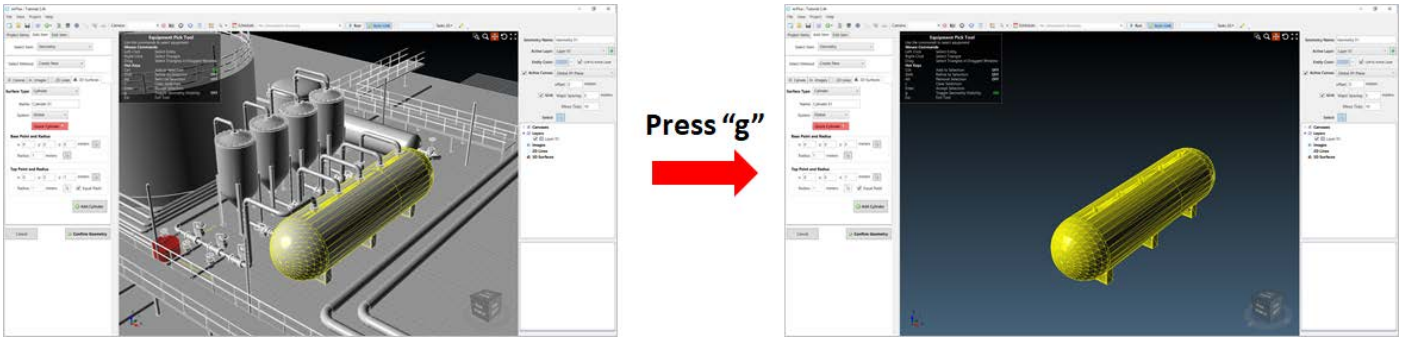




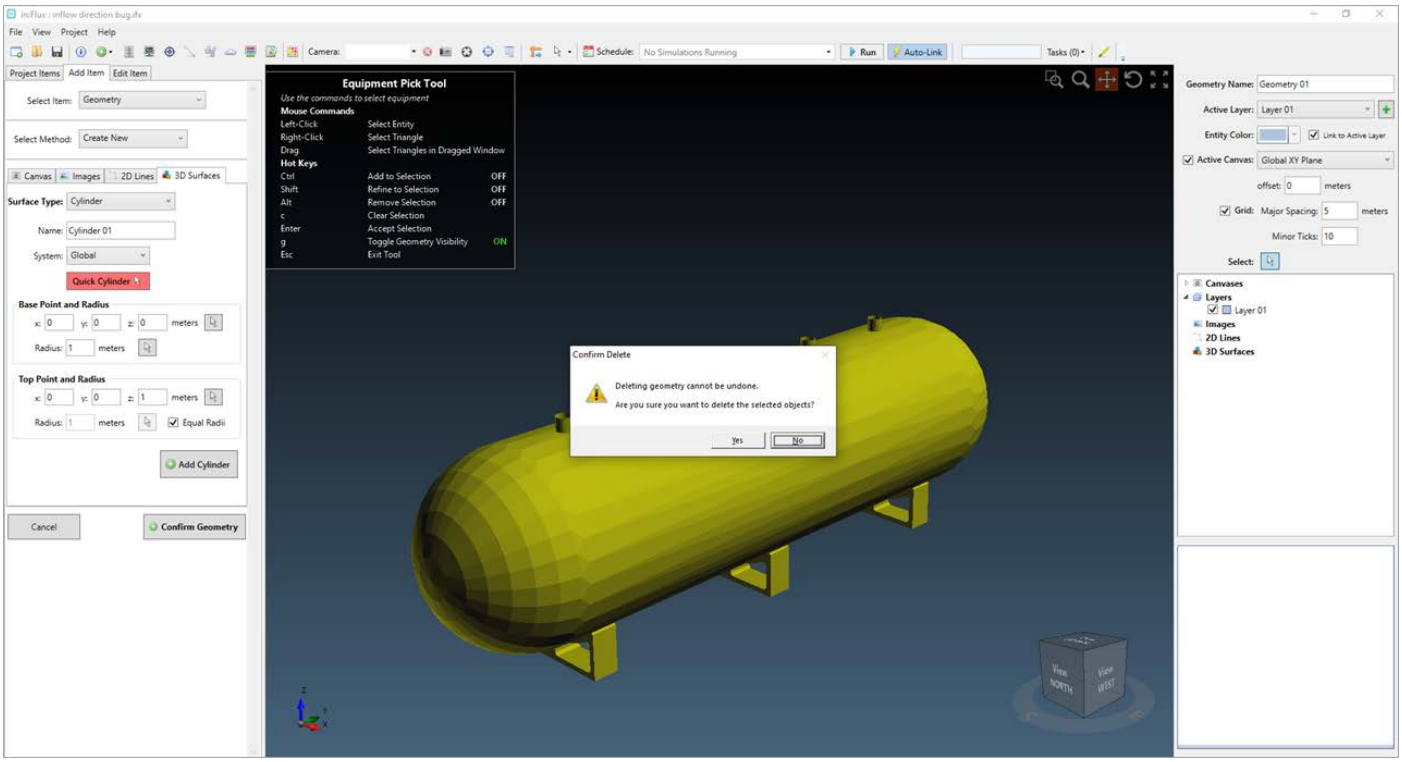
This process can be done multiple times from different view. Below the **Alt+Drag** method was used to remove the upper flanges from the selection by looking at a Side View of the facility.



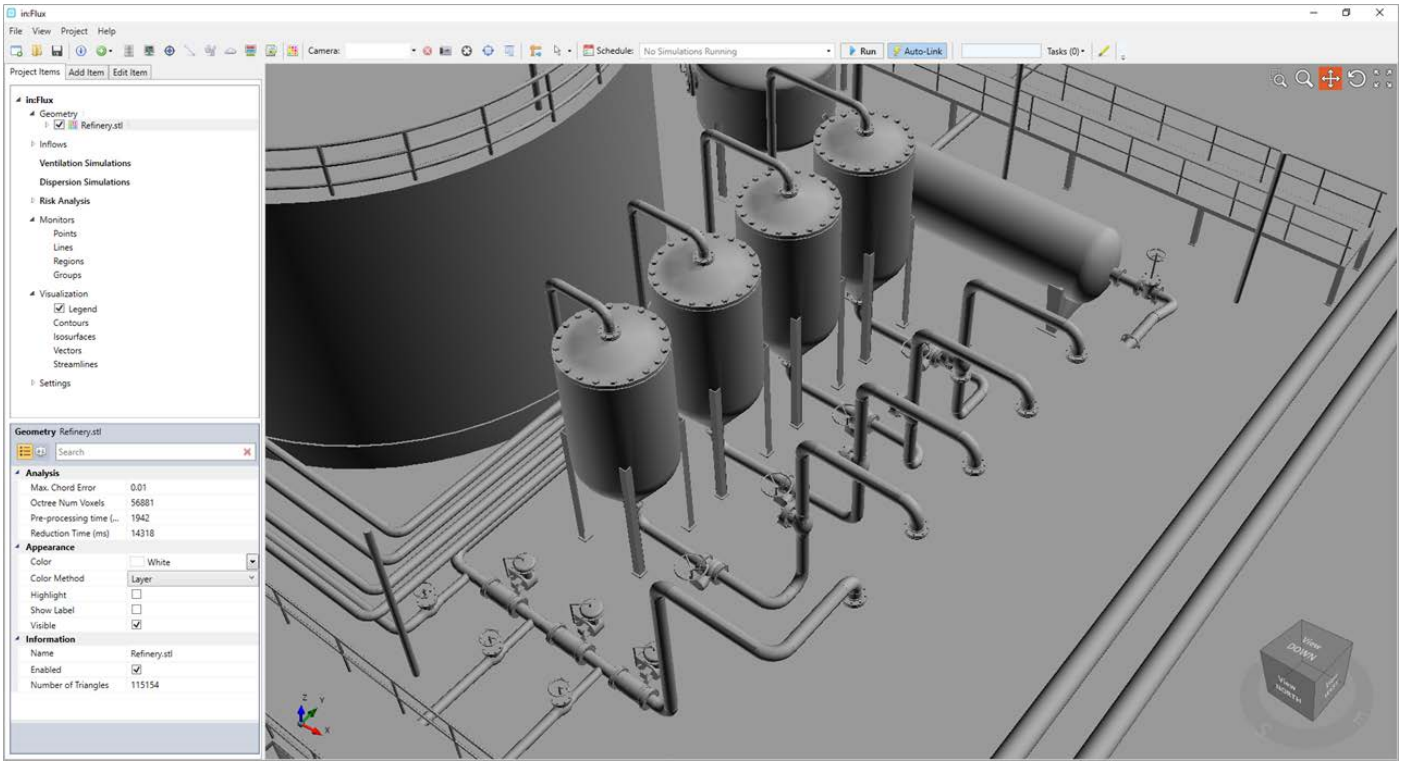
In some cases. geometry pieces are highly complex and difficult to maneuver around. To help with this difficult, or simply to verify that your selection is correct, there is a hot key for toggling geometry visibility. Pressing the "**g**" key, as in the example below, will **visibly** remove the geometry pieces which are not selected on the screen.



Once you are happy with the highlighted section, press the Delete key on the keyboard to remove all of the highlighted triangles from the CAD model. A warning window will appear (shown below), click *Yes* to confirm deletion. Be sure to save your project beforehand as this cannot be undone.



Pressing **Esc** will exit the Equipment Pick Tool and click  at the bottom left of the screen to confirm the updated CAD model.



CAD Geometry Representation and Simplification

All geometry in Detect3D and in:Flux is a collection of triangles. The conversion of complex surfaces into a set of triangles is called "tessellation." Very often, the default settings for which CAD programs export are not appropriate for quick analyses and will lead to overly detailed geometry.

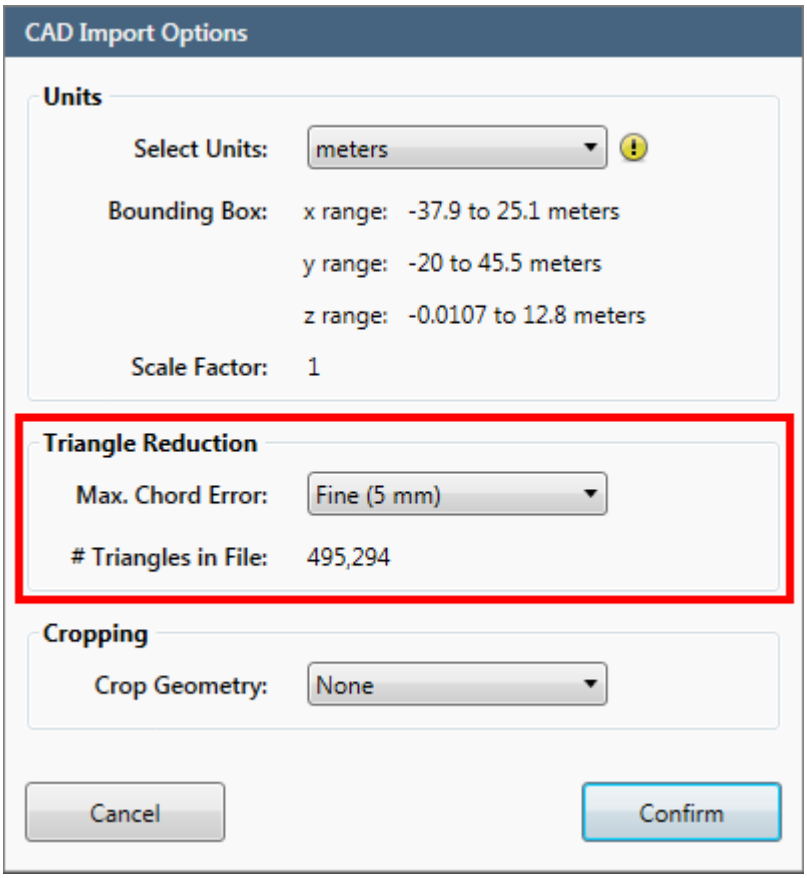
In general, the smaller an object is the less important it becomes to the accuracy of the simulation. It is wasteful for computation time to use 500,000 triangles resolving nuts and bolts on flanges and other superfluous CAD pieces.

Spending time to reduce the number of triangles in the geometry will be time well-spent to significantly increase computation speeds. Note that even If you convert a CAD file using external software, **you have control over how many triangles are created.**

This section will go into how to properly use the triangle reduction capability used in Detect3D and in:Flux.

Insight Triangle Reduction

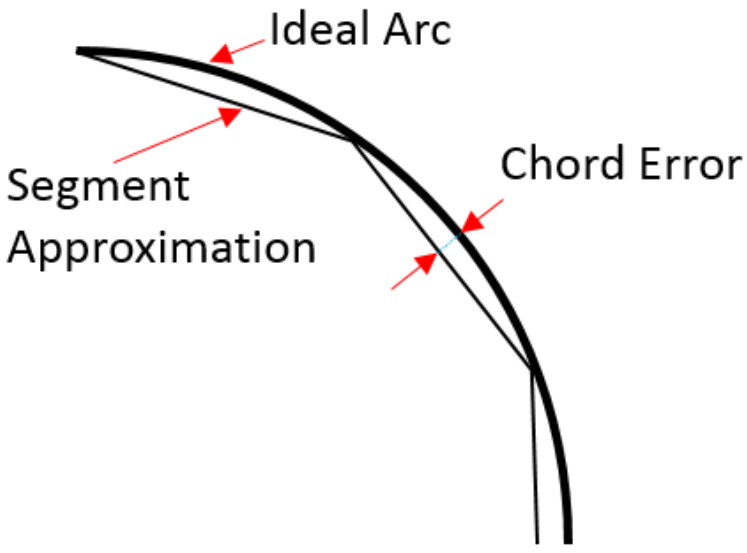
When loading CAD files into Detect3D or in:Flux, the CAD Import Options window (shown below) will appear. The section highlighted in red below controls the Triangle Reduction of the file.



CAD Import Options Window indicating the Triangle Reduction section.

Both Detect3D and in:Flux have a built in geometry simplification algorithm. The algorithm helps when importing CAD files and simplifying the number of triangles which exist in the CAD. The Import Options Window shows an initial calculation of the number of triangles in the file as well as an option to change the *Max. Chord Error.*

The *Max. Chord Error* is analogous to the example shown below. It is a term used to describe the difference between an idealized arc and the approximation using segments. The number of segments is chosen so that the maximum error (distance between the two red triangles) equals the chord error setting.



Pictorial description of Chord Error in relation to an idealized arc

Users can choose one of the 6 options from the dropdown menu for the Max. Chord error: None, Very Fine (2mm), Fine (5mm), Medium (10mm), Coarse (15mm), and Very Coarse (20mm). The mm distance can be thought of as the distance allowed away from the original mesh points (triangle vertices).

Insight Numerics software defaults to different Chord Errors for different numbers of triangles in your CAD file, shown below. You may choose to change the default setting, but it is not recommended.

Triangle Count Range	Default Chord Error
0 - 100,000	None
100,001 - 250,000	2 mm
250,001 - 500,000	5 mm
500,001 - 5M	10 mm
5M +	15 mm

This process is best done only one time when initially loading a CAD file. There is no need to perform the reductions after the initial import as the geometry will be saved in the .d3d file of the project.

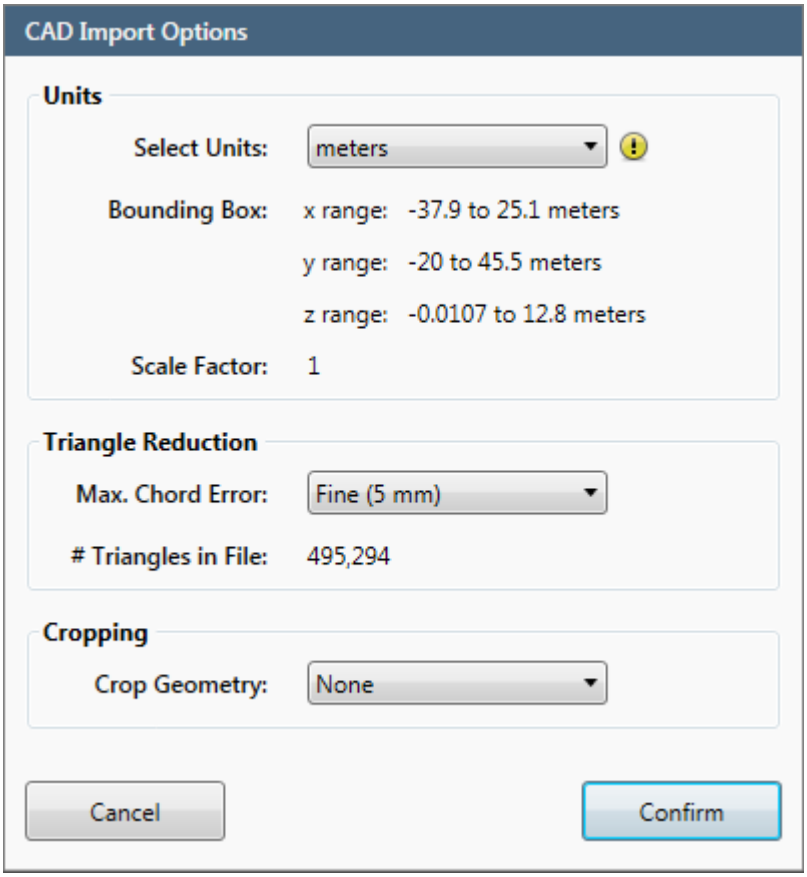
Note: All of the tutorial geometries which are pre-installed have already had triangular reductions performed.

The next section will show a comparison of the Chord Errors using the Refinery.stl file, but feel free to open one of the other pre-installed CAD files and choose the Triangle Reduction option from the Project Menu.

Tessellation Comparison

This section will go over a comparison of the various triangle reduction chord errors to provide a better understanding of the tessellation involved and the importance of reducing the triangle count to reduce the load on the computer's graphics and memory cards.

1. Open up a new project in Detect3D or in:Flux.
2. Locate the Refinery.stl file from the installation directory or from the Help Menu.
3. Load in the Refinery.stl file and wait for the CAD Import Options Window to appear as below



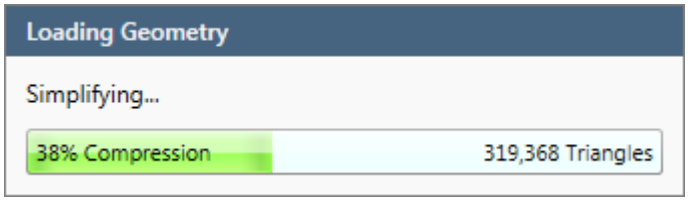
CAD Import Options Window for the Refinery.stl CAD file

4. The units and should be set to **meters** as the bounding box dimensions are appropriate for this project.
5. The Max. Chord Error has already been set to *Fine (5mm)* since there exists less than 500,000 triangles. Choose **None** from the *Max. Chord Error* dropdown menu.
6. We will not be using the Cropping Capability in this section. Leave the Crop Geometry: set to **None.**
7. Click **Confirm** to load the Refinery.stl file.

We will be loading the file 3 more times to compare the triangle reductions.

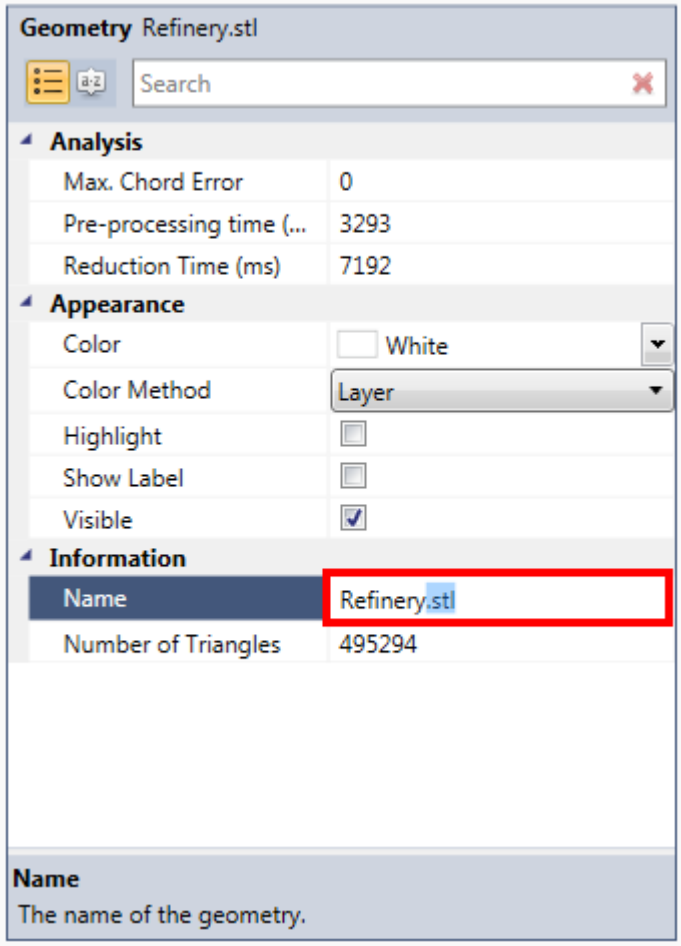
1. Load the Refinery.stl file a second time.
2. Choose **meters** from the Select Units Dropdown Menu
3. Set the *Max. Chord Error* value to **Fine (5 mm)**.
4. Click Confirm.
5. Load one last Refinery.stl file and choose **Medium (10mm)** as the Max Chord Error.
6. Click Confirm to load the last geometry file for Tutorial 5.

Notice the splash window that appears while the triangle reduction is running (see below image). The percent compression is displayed on the left while the new number of triangles is displayed on the right. Note that many CAD files have 1000s of superfluous triangles representing mm level differences (sometimes smaller) so there is no need to worry that 80% or higher has been compressed as it will soon be described that a high level of accuracy is still maintained in the CAD.

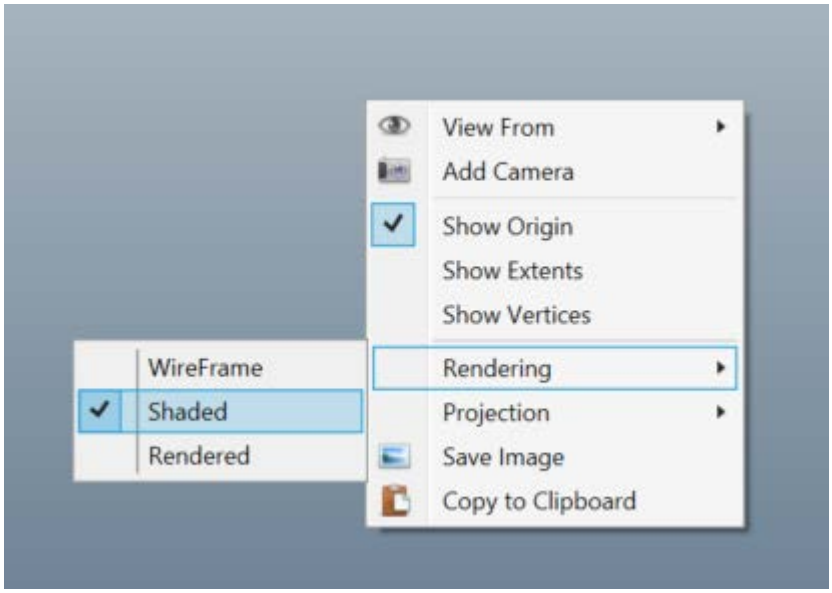


Loading Geometry window while triangle reduction is calculating.

You should now have 3 *Refinery.stl* files loaded
1. Change the name of the first geometry loaded to **Refinery - Original** by selecting it from the Project Items Tree and editing the *Name* text field under the *Information* header, shown below.

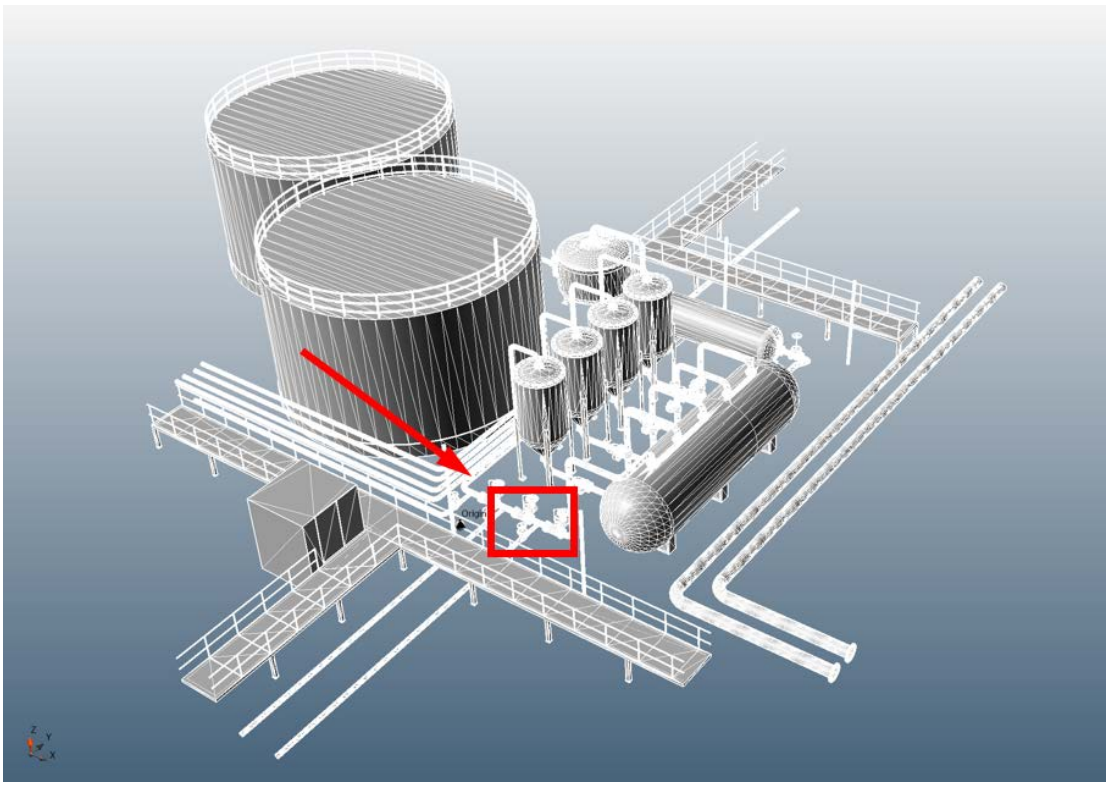


2. Change the name of the second geometry to **Refinery - 5mm**.
3. Change the name of the third geometry to **Refinery - 10mm.**
4. Turn off the visibility of the *Refinery - 10mm* and *Refinery - 20mm* by deselecting the checkboxes next to the names in the Project Items Tree.
5. In order to see the triangles of the CAD geometries we need to change the rendering of the Viewport Window. To do this right click anywhere in the Viewport Window and under the *Rendering* section select the **Shaded** option (shown below). You may also select the *Shaded* option for rendering from the View Menu on the toolbar.



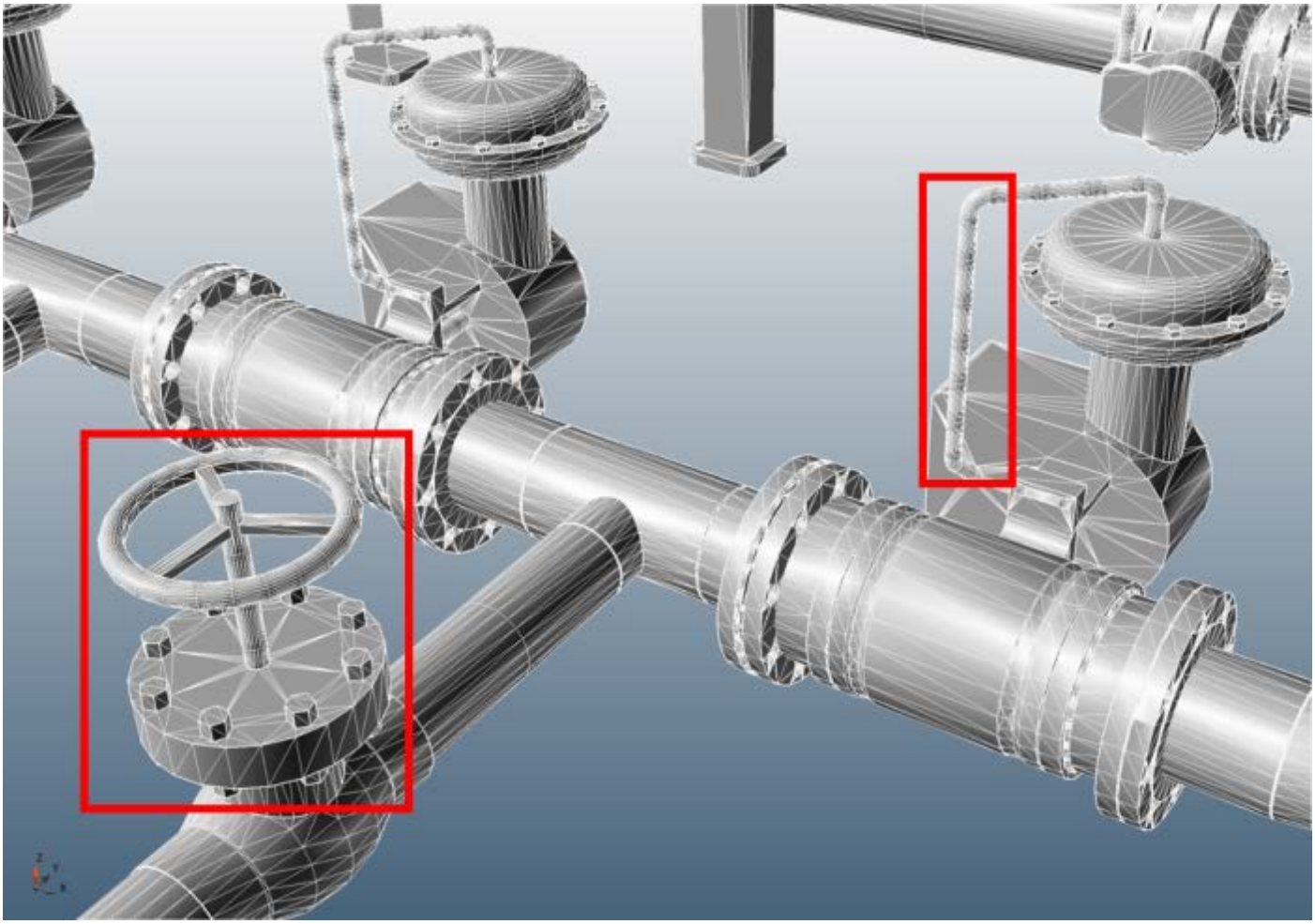
Selection of the Shaded rendering for the viewport window.

Once the rendering has been changed zoom into the boxed region shown:

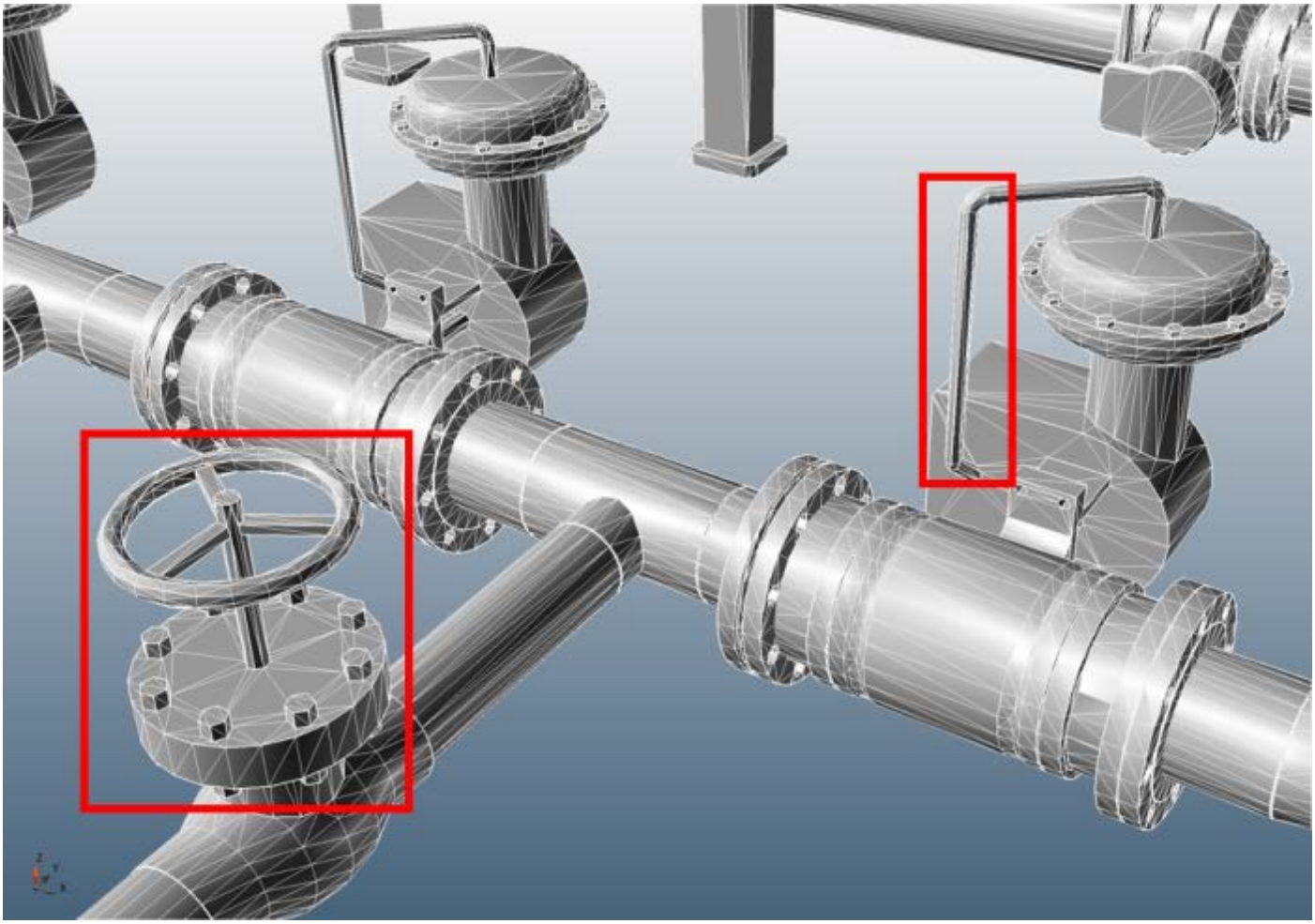


Region to zoom into on Refinery geometry.

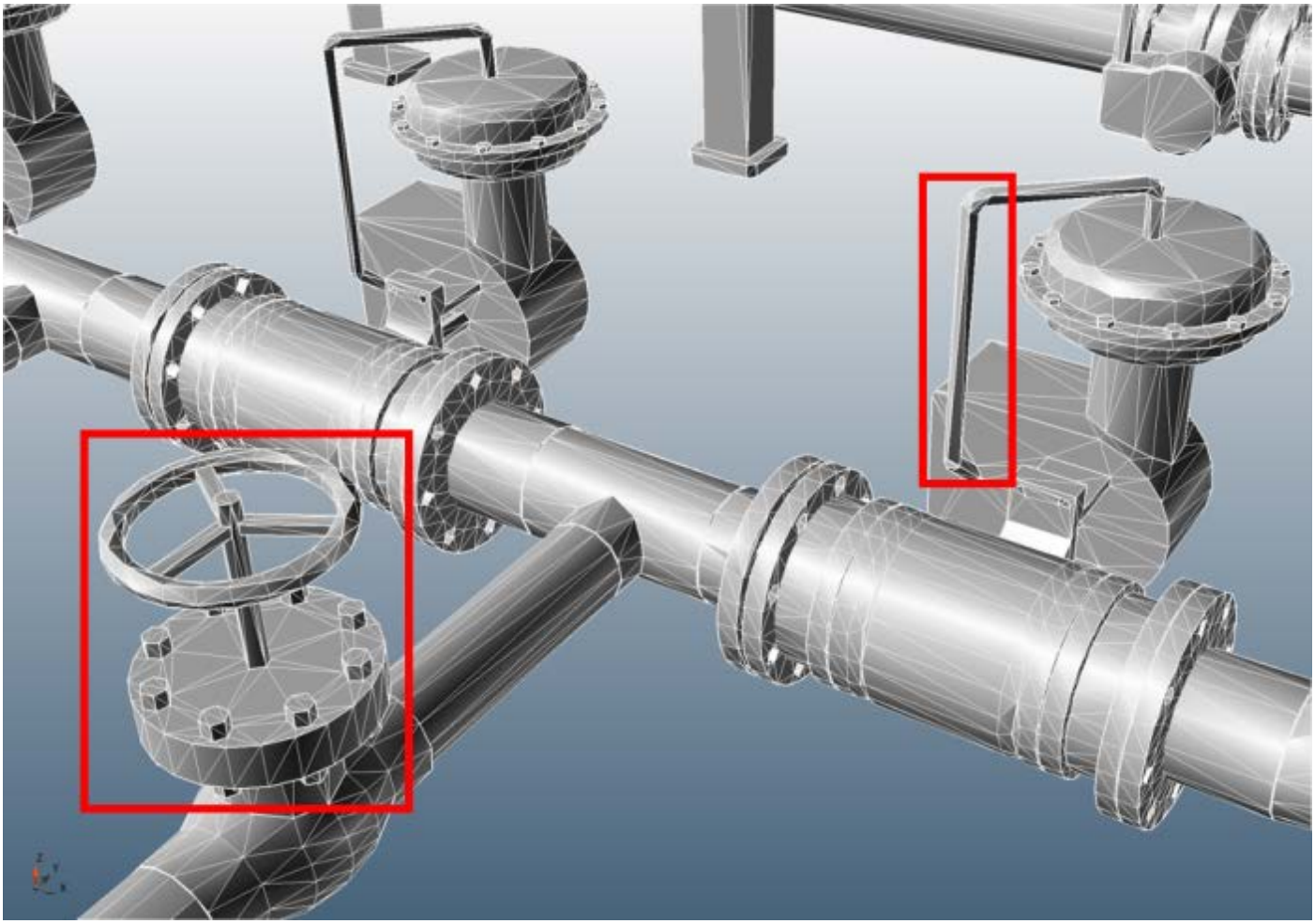
The next three images below show how triangles are used to represent pipe work and valves in a geometry as well as the differences between a coarse tessellation (the 20mm triangle reduction) and a refined one (the original CAD file). You can view the same results as the screenshots below by zooming into the region indicated in Image 18 and toggling the visibility of each Refinery geometry.



Refinery - Original (495,294 Triangles)*:* The above image indicates two regions which are made up of several hundred triangles. Too many triangles have been used to represent these small regions which are almost insignificant for fire and gas mapping purposes. The following two images will show how even with less triangles you can still accurately represent pipes and values.



Refinery - 5mm (171,160 triangles): The number of triangles has been reduced to 35% of the original. The geometry is still well defined with several triangles make up the bolts and small pipes, a larger chord error would be better for this CAD file.



Refinery - 10mm (118,682 triangles): The 10mm triangle reduction is the recommended chord error for CAD files of more than 500,000 triangles. in:Flux reduced the triangles of the geometry to 24% of the original amount. Even though the triangles are coarse (could still be coarser) they still accurately represent the refinery geometry and will allow for much shorter analysis times.

NOTE: to reduce the load on the graphics card, users have the option to limit the size of the small triangles **displayed** by the graphics card. To change this setting:

1. Select Application Options from the Project Items Tree
2. Change the value of the Small Triangles Size to **10** (the default is set to 2).
3. Selecting Show Wireframe On Motion also assists in reducing the load on the graphics card.

Conclusions

Geometry Tessellation
CAD files are represented by thousands and sometimes millions of triangles. The CAD files loaded into Detect3D and in:Flux can be compressed to facilitate quick analyses. A default selection is made when loading CAD files and is dependent on the number of triangles in the CAD.

Notes on Geometries in Detect3D and in:Flux
• Remember that you can create geometry pieces using the *CAD Creation Tool*
• For most geometries, even complex ones, the triangle count should not be greater than 3-4 million triangles. If you have geometries that are on the range of 5 million triangles or more, take a look at changing the *Max. Chord Error* mentioned earlier or attempt to separate or edit the CAD file.
• Be knowledgeable about using third party triangle or mesh reduction capabilities and the options they have for exporting.

Once you have found appropriate settings for exporting and importing to and from Detect3D and in:Flux, you can re-use them on all projects.

Contact Information

Insight Numerics directly provides full technical support and updates for its software products. If you have purchased or leased a license from Insight Numerics, you are entitled to unlimited technical support, maintenance, and version updates. Please use the contact information below if you have questions about the technical support included with your license purchase.

As the sole developers of in:Flux and Detect3D, all updates are issued and maintained by our head office in Boston, Massachusetts. If you have any comments or would like to request additional capabilities to be added to any of our software products, email us your ideas and we'll be happy to discuss its implementation into the next software update.

Company website:
http://www.insightnumerics.com

Support e-mail:
support@insightnumerics.com

Boston Headquarters Phone Number:
+1 - 617 - 657 - 4708
Office Hours are 8am to 6pm (EST) Monday through Friday