RhinoCFD Tutorial - Convert Revit files into PHOENICS/RhinoCFD compatible formats
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Convert a Revit model to RhinoCFD / PHOENICS compatible file

Revit models can be successfully imported into PHOENICS and RhinoCFD if the appropriate procedure is followed when exporting the model from its native format.

Key parameters are to ensure that the solids from Revit are exported as **ACIS solids** and that some tolerance settings are adjusted in Rhino3D. Additional optimisations during export can reduce the complexity of the pre-processing of the case significantly. These optimisations can be done via Rhino3D – used here for demo purposes.
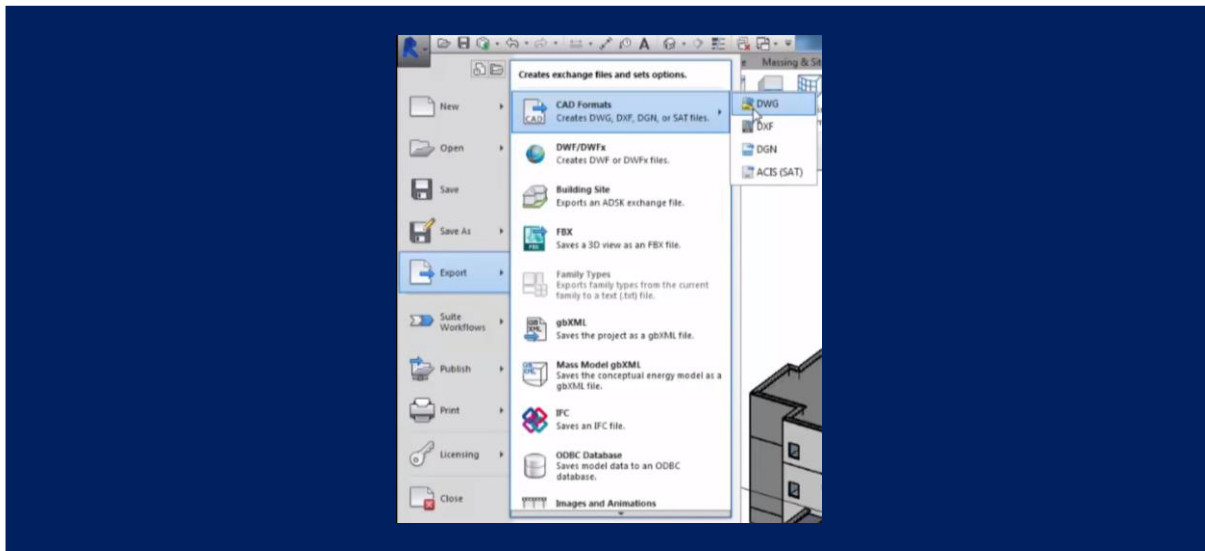
This tutorial describes the steps needed.
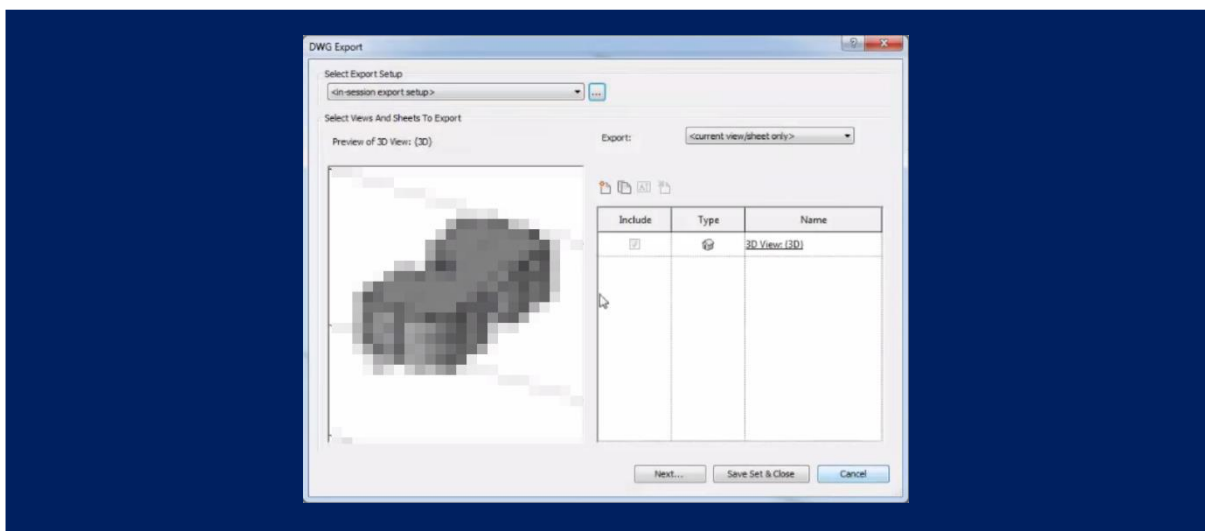
Exporting from Revit:

Step 1

Open your Revit file and go to:
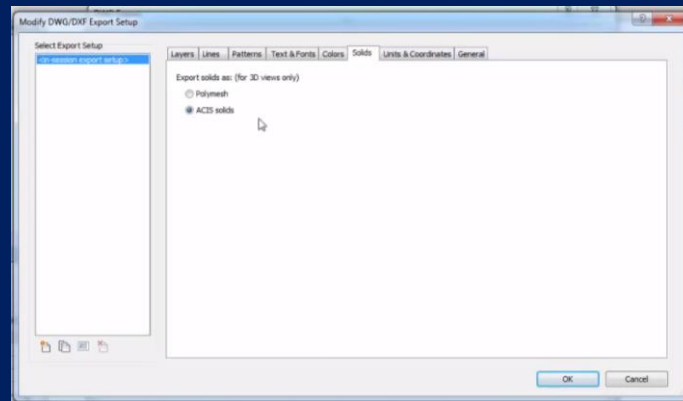
- File > Export > CAD Formats > DWG

Step 2
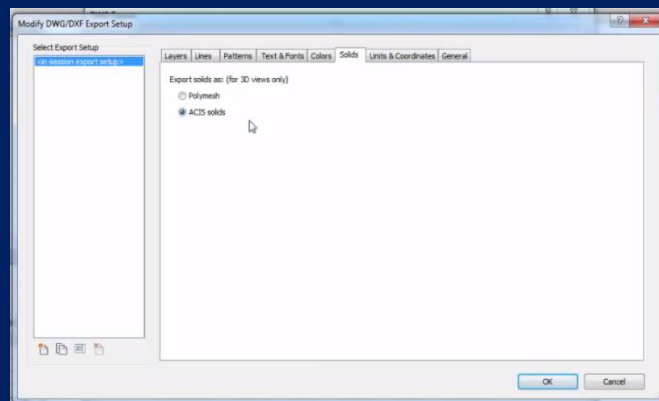

With the "DWG Export" window open, click on Export Setup configuration button.
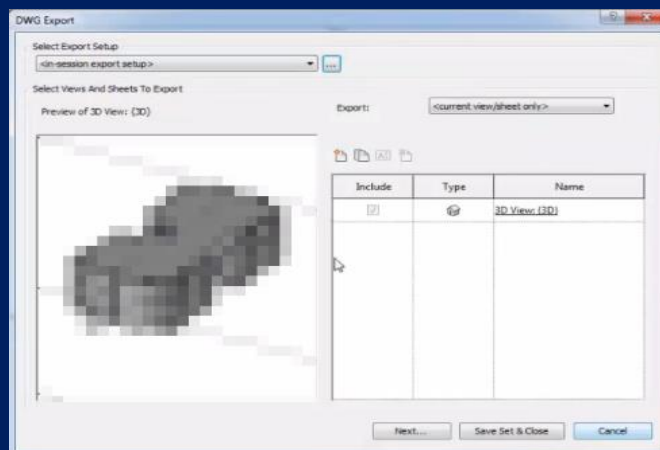
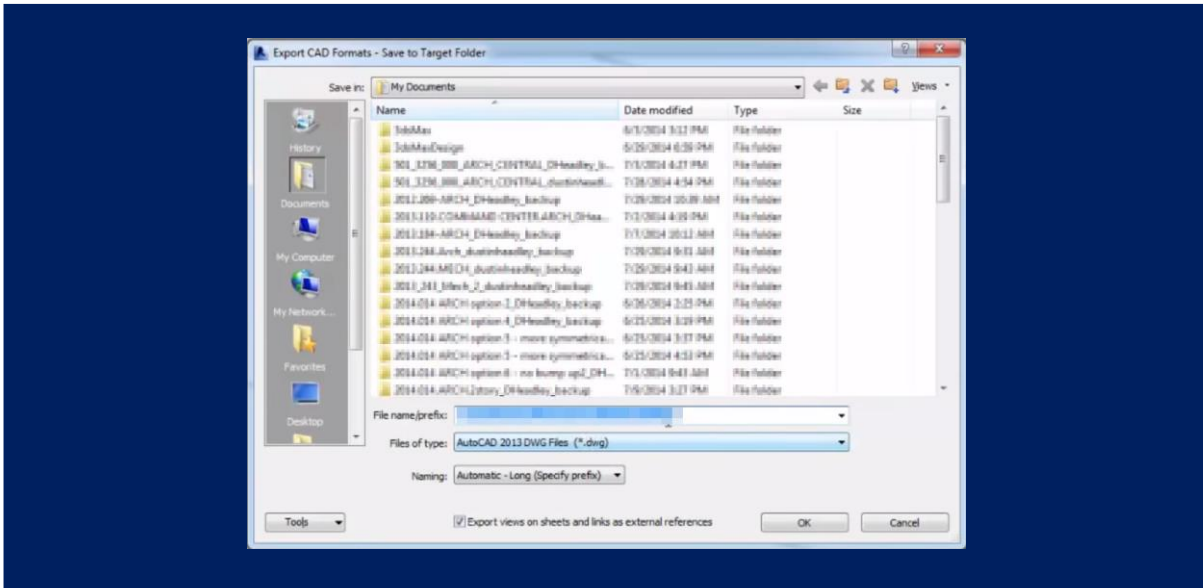With the "Modify DWG/DXF Export Setup" window open navigate to the Solids tab and select **ACIS solids.**

Press "OK" on "Modify DWG/DXF Export Setup" window

Press "Next" on "DWG Export" window
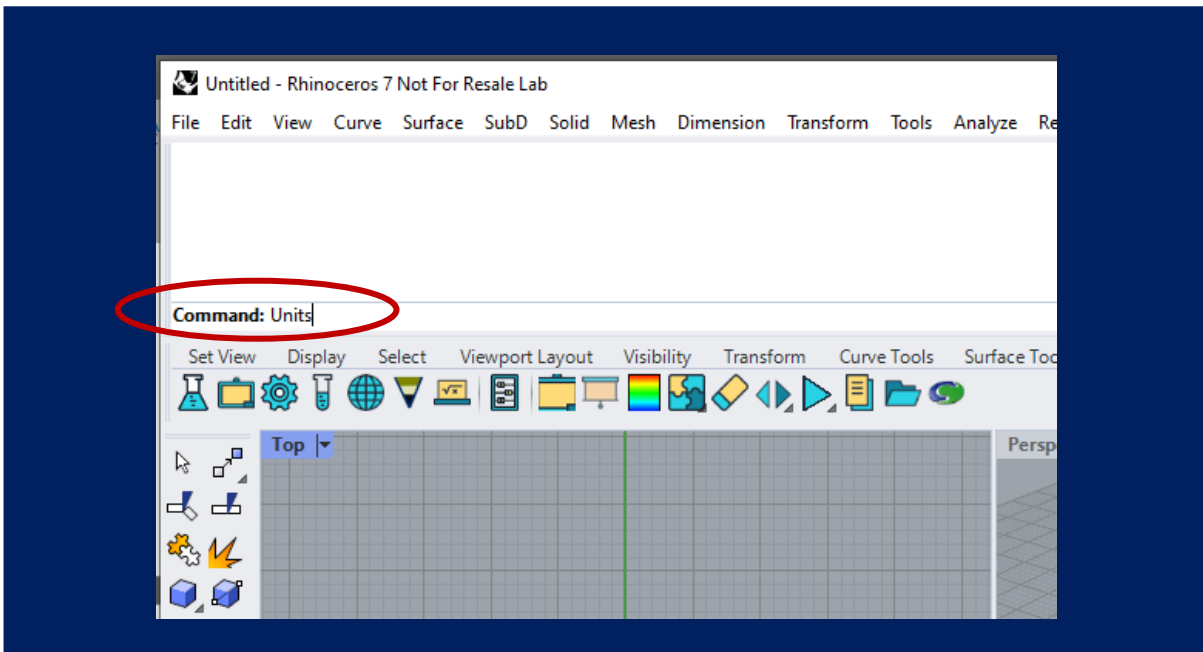
With the "Export CAD" widow open press "OK".

At this point the .DWG file is capable of being imported into either RhinoCFD or PHOENICS. Using Rhino, we can further optimize the geometry for easier pre-processing and potentially creating a better mesh for the CFD case.
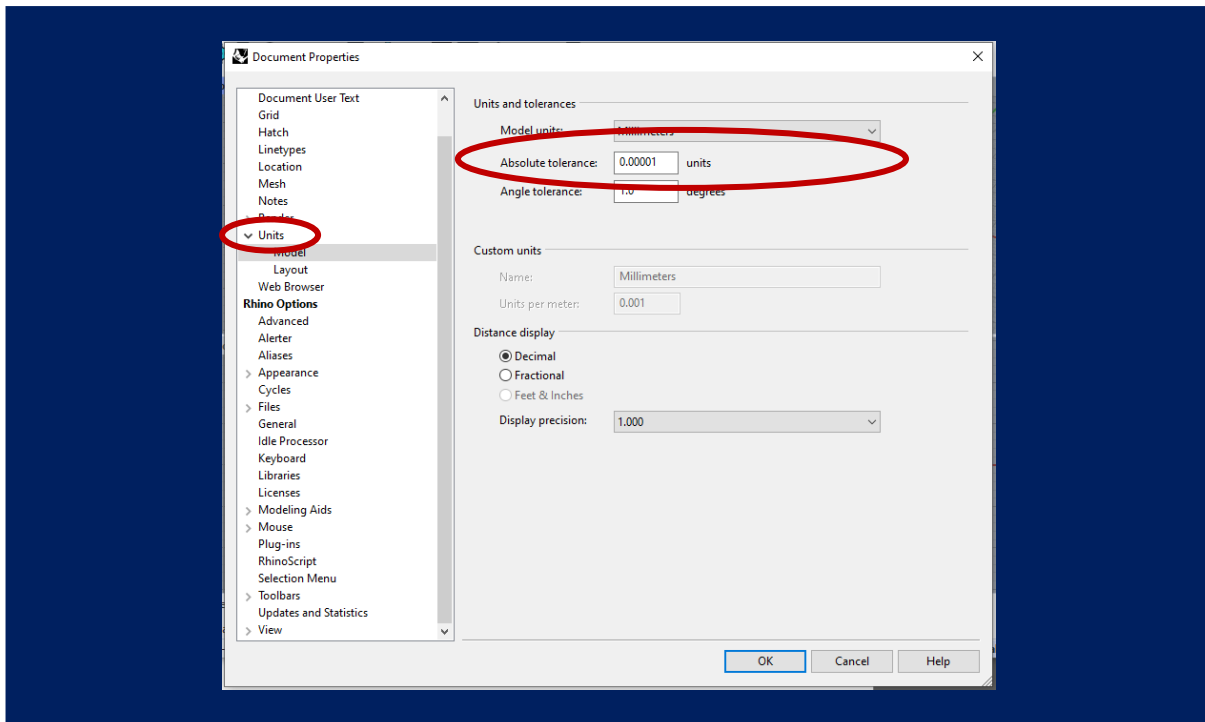
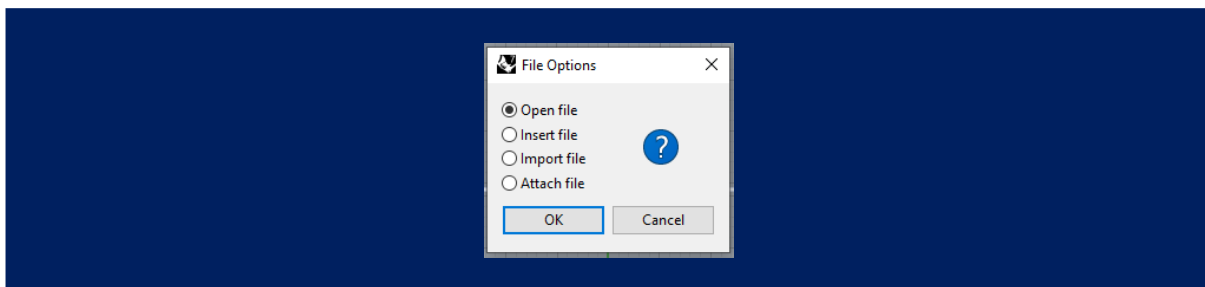The optimisations are discussed in the last section of the tutorial.

In Rhino type in "Units" in the command prompt
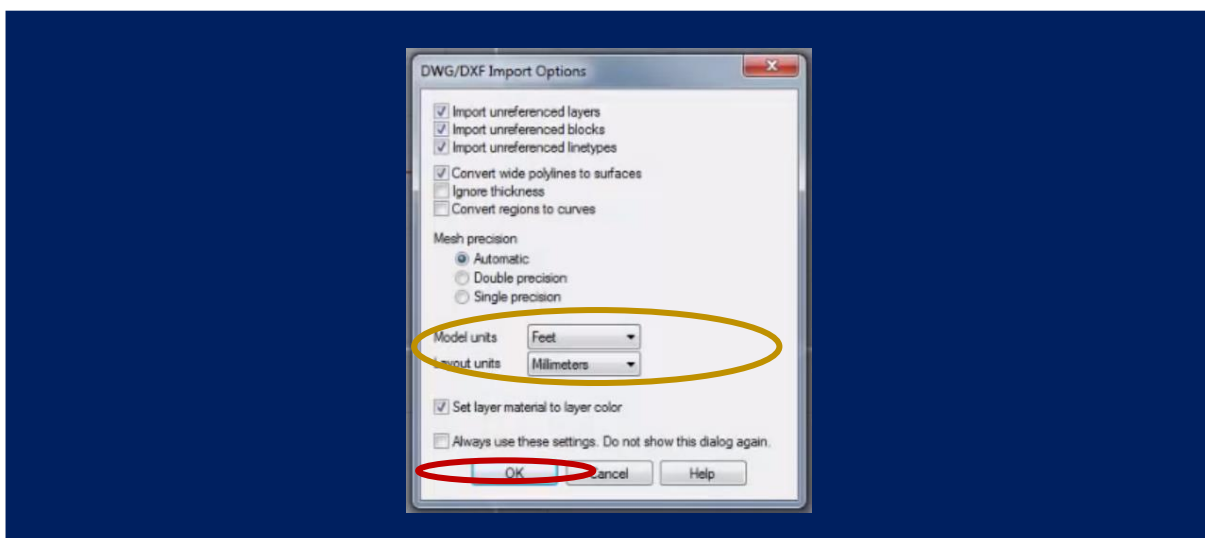
Lower the Absolute tolerance by a few decimals.
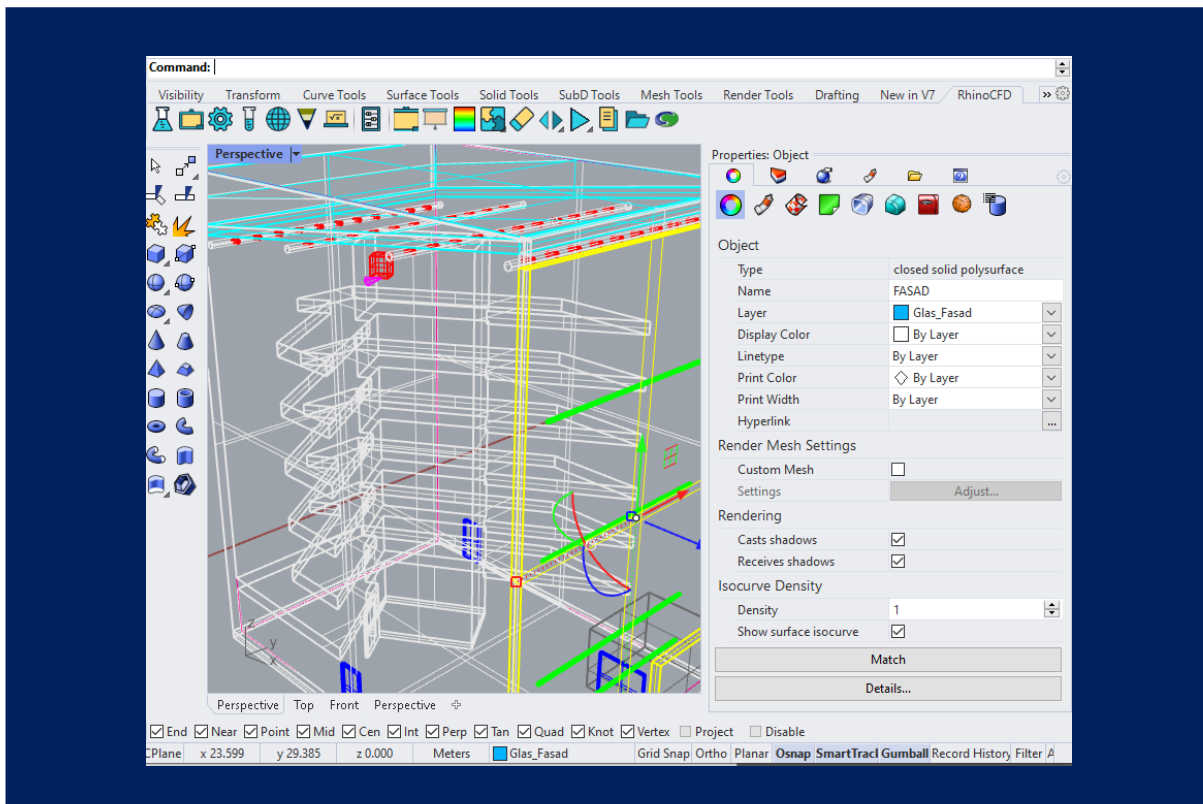
We can now drag+n+drop our *.dwg file into Rhino

Select "Open file" and press OK.

Select the Model and Layout units according to your CAD files and press OK.

If we now click on an object/part from the Revit model we will see it is detected as a closed solid polysurface.

These objects can be successfully read by RhinoCFD.

Important Optimisations

In Revit

Revit files usually have many layers of information, not all of which will be necessary for the CFD analysis. It is therefore good practice to trim the Revit model before exporting.

In Rhino

When passing objects to the RhinoCFD plugin it can be helpful to separate the geometry into blocks of interest. This will reduce the complexity of the CFD mesh.

For example:

- Walls (and windows and others misc included within the walls) can be blocked into a single object if they are simply acting a containment for the room.
- In cityscape models, the buildings that are not of interest can all be blocked as a single Rhino object

To block objects in Rhino:

- select the objects
- type "Block" in the command prompt
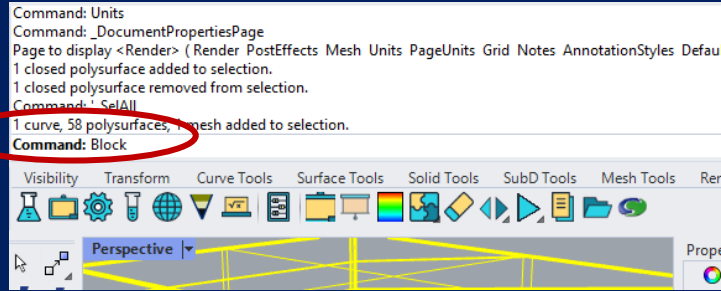
- Set base point to 0, give it a name and press OK