# Introduction

This technical note describes the differences between the properties for height and how to apply the properties to PCB Editor symbols either directly in the library or via the front end tools (Capture, Capture CIS, DE CIS). Since the introduction of the 16.3 release, PCB Editor has a 3D view capability. This will either show the height defined on the PLACE_BOUND shape or display the actual STEP model that is attached to the PCB Footprint if it has been defined. Users have always had the ability to define a height property whether it be a single height or a min / max type. This information can then be transferred to a 3D mechanical tool to create accurate 3D models if the STEP model is not assigned. You can also use Keepout areas to restrict certain height components from areas on the PCB.

Height Definition in PCB Editor

To apply a height property to a PCB Editor symbol it is best to define this whilst creating the symbol. Every symbol must have a PLACE_BOUND_TOP area which defines the keepout area for the component. This area is used during placement to stop placing components over other components. If you do not add this area it is added automatically based on the extent of the pins and outlines drawn. It is recommended that you add this area when generating the symbol. Once the shape is added use Setup > Areas > Package Height then LMB (left mouse button) the shape, it will highlight, the Options Pane will display the following:-
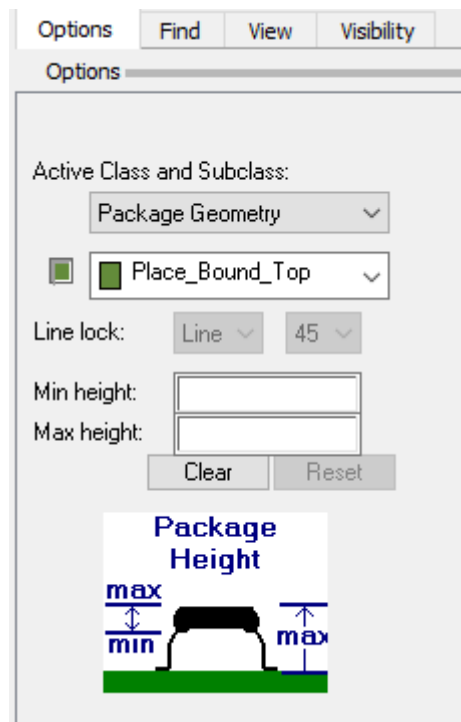
Set the Min (If required) and Max heights. If you only have one height only set the Max height. The min height is normally used if the component does not sit on the surface of the PCB directly. This gives users the option to place components underneath other components for example a DIP carrier has space below allowing surface mount components to be placed underneath the part. This is not a recommended design practice and can have

implications with Assembly. Once the heights have been defined use RMB (right mouse button) Done or F6. Save the symbol and the height will be stored for future use.

The Following picture describes the min, max height definitions with a typical usage model of a Package Keepout with a height defined.
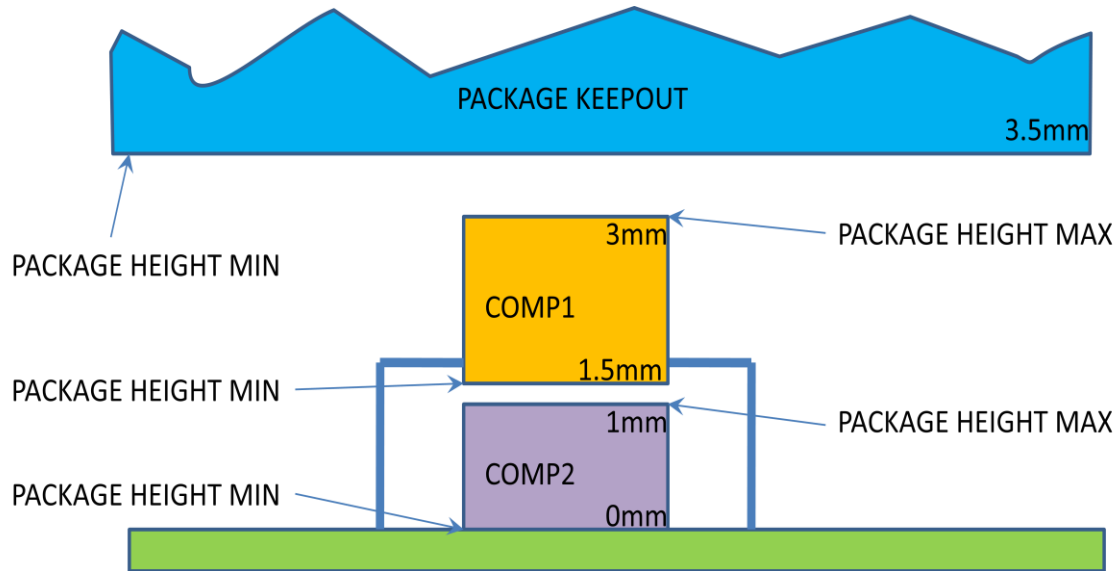
Note: If the Place_Bound_Shape (Top/Bottom) does not have a height defined then the shape will use the default component height specified from Setup – Design Parameters – Design Tab – Default Symbol Height.

Height Definition for a Keepout Area

To define a height restricted keepout area use Shape – Rectangular, Polygon or Circular then set the Active Class to Package Keepout, subclass to the required TOP, BOTTOM or ALL layer then draw the area. This creates a default Package Keepout area that will show a DRC if any component regardless of height is placed within it. You can define a height property for the Package Keepout area using the same method for symbols. Setup – Areas – Package Height(Allegro) or Outline – Package Height (OrCAD) then select the shape with a left click, then define the min and max height accordingly based on the Options Pane. The figure above shows a good example if you want components below a certain height under a keepout area (like a cover for example).

Height Definition from OrCAD Capture / DE CIS.

You can use the front end tools to apply the height properties into PCB Editor. This way you can use the same PCB footprint symbol (i.e. 0603) but have differing heights depending on the individual part (i.e. 0603 resistor height 1mm, for a 0603 capacitor height 1.5mm). It is IMPORTANT to note that your PCB symbols do NOT need heights defined.

To define this in Capture select the part and RMB Edit Properties, Add a New Property e.g. Component_Height and for the value specify the symbol height e.g. 2. The units are used on import into PCB Editor, so if your units were inches the height of this component would be 2 inches. When using OrCAD Capture CIS or DE CIS add a new column to the database Component_Height and add the values according to the symbols.

Before you create the netlist edit the allegro.cfg file. To do this Select the dsn filename in the Capture Project window, Tools – Create Netlist,
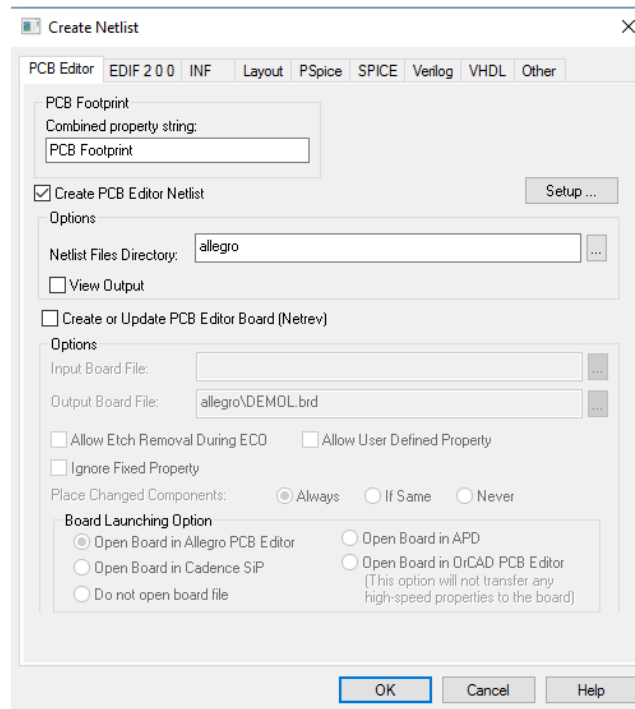

Select the Setup button
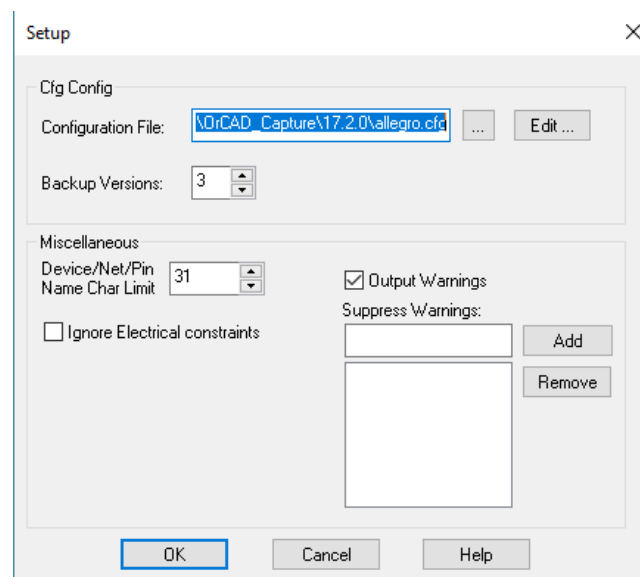
Select the Edit button and add a new Component Definition Property Component_Height=HEIGHT. Save the allegro.cfg file and OK the setup dialog, then OK to create the netlist.

There is an option in OrCAD Capture that will allow you to store the allegro.cfg file in a user defined location. Use Options – Preferences – More Preferences, then under Extended Preferences select Configuration Files then, Edit the location for the allegro.cfg file. This is stored as part of the capture.ini file and will be set for future use.
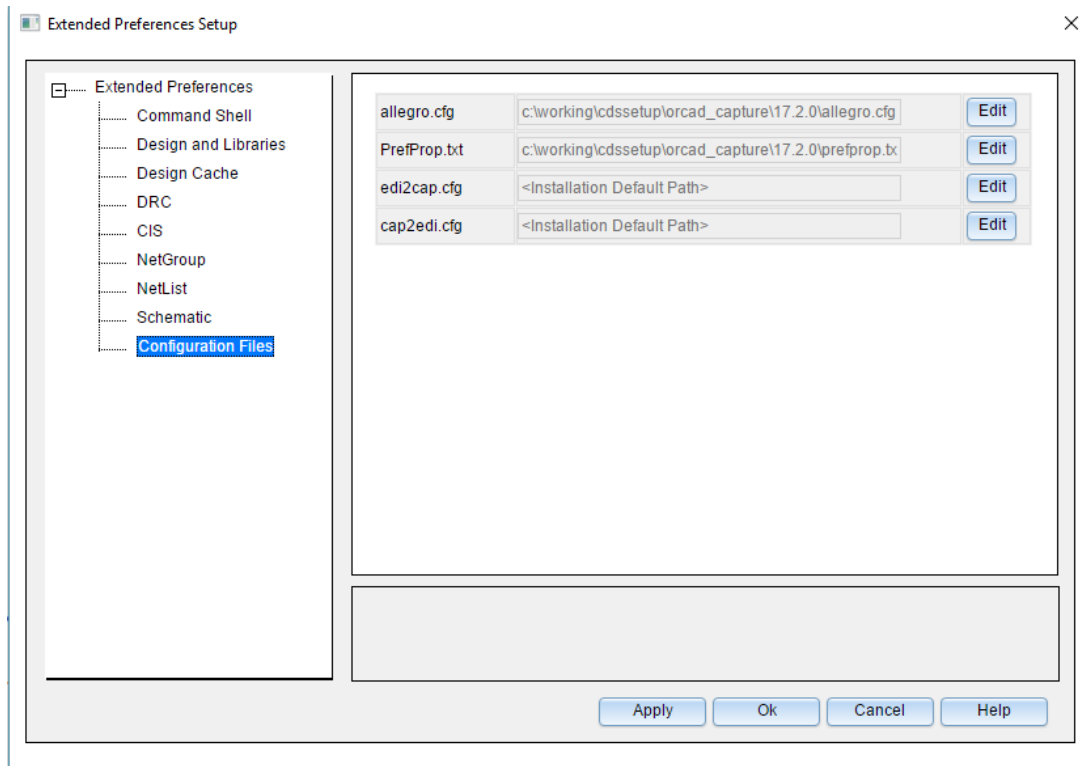
Import your netlist into PCB Editor and place your components. The HEIGHT property defined in Capture / CIS populates the PACKAGE_HEIGHT_MAX and HEIGHT if you use Show Element on a symbol. To see the components in a 3D view use View > 3D View (Allegro) or Display > 3D View (OrCAD). If a STEP model is mapped to the symbol you will see this instead of the PLACE_BOUND shape.
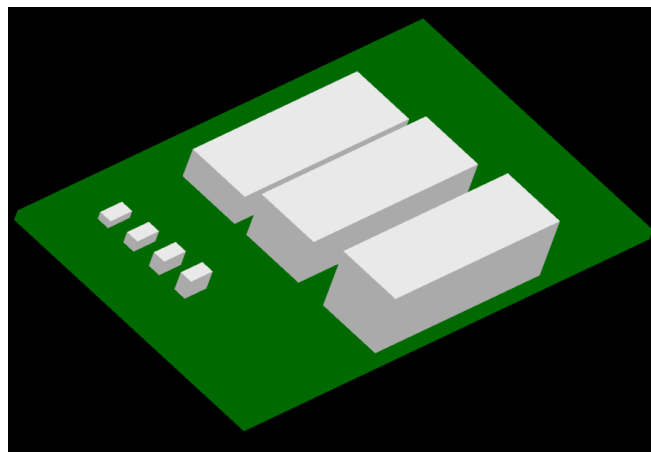
If you are exporting an IDF file, File > Export > IDF (Allegro) or Export > MCAD > IDF (OrCAD) for use with an external 3D Mechanical tool like AutoDESK Inventor then you need to complete the following.

There is a user preference Setup > User Preferences > Interfaces > IDF called idf_ignore_comp_height. Check this property then when you export your IDF file and the symbol based height property is used.

Creating a report of Heights for Components.

You can use the text below to create an extracta text report file (save as filename.txt) so that the Report command will write a report. To use this, save the text file to a location like the same directory as the board file or a network location, then set the textpath User Preference (Setup > User Preferences > Paths > Config) to point to this location. Then use Tools > Reports (Allegro) or Export > Reports (OrCAD), browse to the text file or choose it from the list of reports, and then click on Report to run the extracta command and write the report.

```
GEOMETRY
REFDES!=""
CLASS="PACKAGE GEOMETRY"
SUBCLASS="PLACE_BOUND_TOP"
OR
SUBCLASS="PLACE_BOUND_BOTTOM"
REFDES
REFDES_SORT
SUBCLASS
GEO_PACKAGE_HEIGHT_MIN
GEO_PACKAGE_HEIGHT_MAX
END
```

Note:- The PLACEBOUND of components will be extracted as line segments that make up the place bound shape meaning that you will get multiple entries per component.