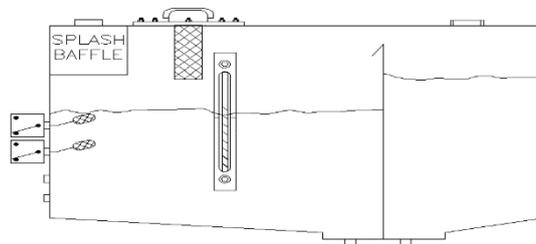


# Fluid (Hydraulic/Pneumatic) / P&ID Symbol Creation

## AutoCAD Electrical

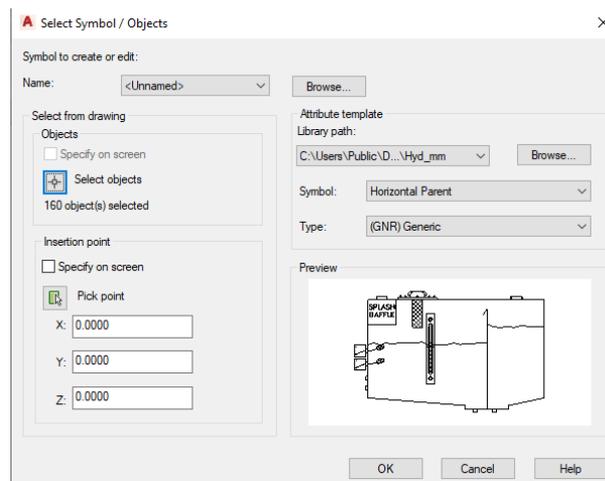
Pneumatic, hydraulic & P&ID symbols contain additional attributes to electrical symbols. We will create a new symbol and explain the additional attributes and their purpose. In this instance we are showing an example of a hydraulic component.

Draw using normal AutoCAD® commands a reservoir tank. It does not have to be as detailed as shown:

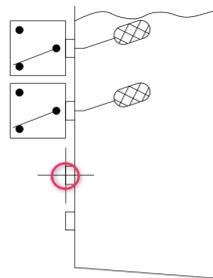


Select *Schematic* ribbon tab > *Other Tools* panel >  **Symbol Builder** ▾

Select  **Select objects** button and window around the graphics.



Select the insertion point  and pick the first primary connection e.g.



Define the following:

The 'Attribute template' dialog box is shown. It has a 'Library path' field with a dropdown menu set to 'C:\Users\Public\D...\Hyd\_mm' and a 'Browse...' button. Below this, there are two more dropdown menus: 'Symbol' set to 'Horizontal Parent' and 'Type' set to '(GMR) Generic'.

Type This is not the intelligence of the symbol but purely the attribute template that your new symbol will use (placement of attributes).

The preview of the dialogue will now look like the following:

The 'Select Symbol / Objects' dialog box is shown. It has a 'Name' field set to '<Unnamed>' and a 'Browse...' button. Below this, there are two main sections: 'Select from drawing' and 'Attribute template'. The 'Select from drawing' section has a 'Specify on screen' checkbox, a 'Select objects' button, and '160 object(s) selected'. The 'Attribute template' section has a 'Library path' field set to 'C:\Users\Public\D...\Hyd\_mm', a 'Browse...' button, and two dropdown menus: 'Symbol' set to 'Horizontal Parent' and 'Type' set to '(GMR) Generic'. At the bottom, there is a 'Preview' window showing a technical drawing of a mechanical assembly. Below the preview, there are 'OK', 'Cancel', and 'Help' buttons.

Select  to *Open* the symbol within the *Block Editor*

A list of available attributes will appear in the *Symbol Builder Attribute Editor* dialogue.

Required	
TAG1	TK
MFG	
CAT	
ASSYCODE	
FAMILY	TK
DESC1	
DESC2	
DESC3	
INST	
LOC	
XREF	
WDTAGALT	
WDTYPE	HY
WDBLKNAM	_HYD-TNK

Enter the following:

TAG1	TK	(Component tag prefix)
FAMILY	TK	(Component family type used for filtering on the specific catalogue table)
WDTYPE	HY	(Component filter used for reports:

PN	=	Pneumatic
HY	=	Hydraulic
PI	=	Process & Instrumentation
1-	=	Single line

You can create your own types which are then shown in the filtering of reports e.g. BD = Block diagram).

WDBLKNAM    \_HYD-TNK    (Catalogue table override to specifically go to the HYD-TNK catalogue table instead).

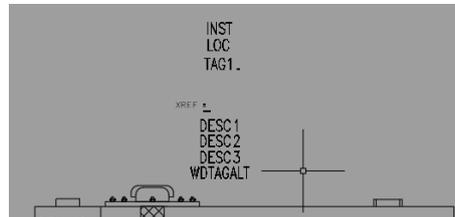
Note:    New groups can be created specifically for your products e.g. \_PID-MOT

WDTAGALT    (Alternative component tag used when a component has two TAG ID's e.g. solenoid and a valve)

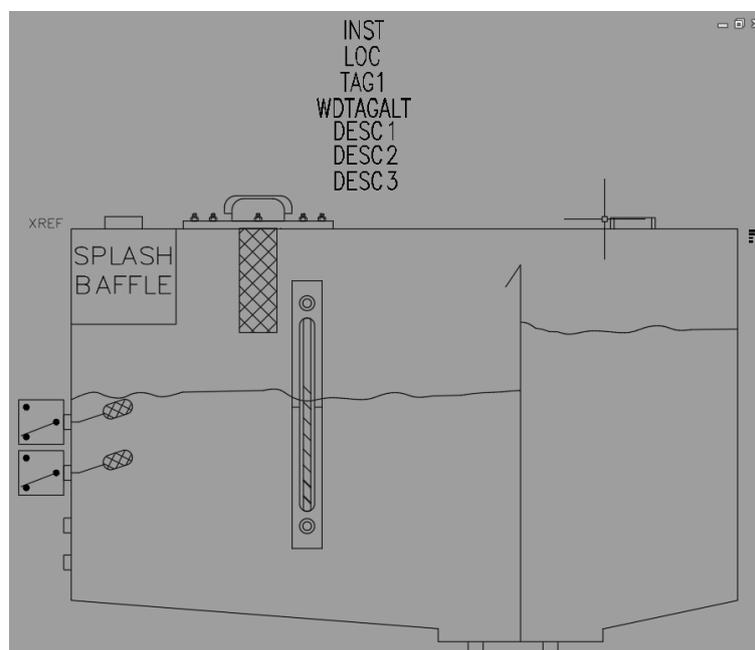
The user can create their own families for different types of components. In order to stay consistent with other schematic types such as pneumatic and electrical, we would recommend that you read the help section titled: [Symbol Naming Conventions](#).

If you do use the WDBLKNAM override when creating symbols, then you could create specific tables for hydraulic components similar to the one defined above.

Highlight all the required attributes using *Shift* and/or *Ctrl* and select  *Insert Attributes*.



Reposition the attributes as necessary:



The WDTYPE, WDBLKNAM, FAMILY, MFG & CAT attributes are *Invisible* and small as a default. These can be changed if you wish using *Properties*.

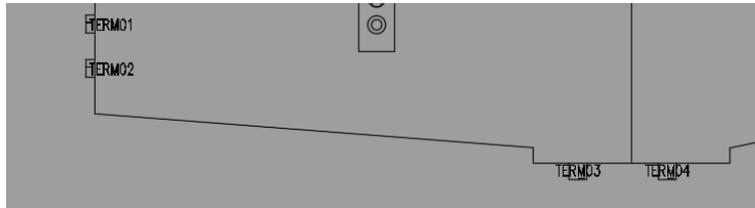
### Adding the Connection points

Under the section *Wire Connections* add the appropriate wire connections.

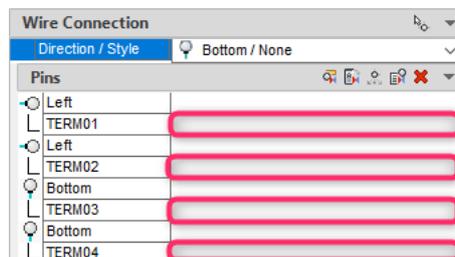


Left click on the appropriate Direction / Style option to  *Insert Wire Connection*.

Repeat for all remaining connection points required.



If the connections have specific identifications, then these can be entered:



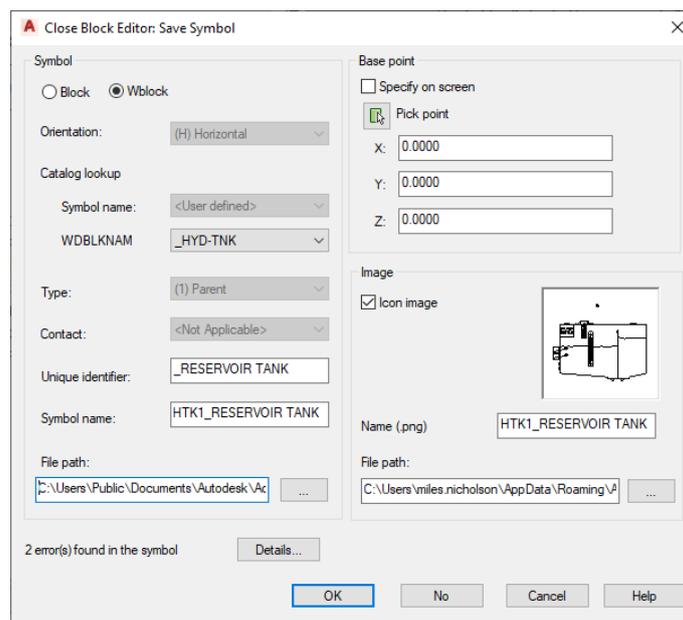
If you wish to review your symbol, select the *Block Editor* ribbon tab > *Open/Save* panel >



To close the test block preview, select



Select



Change the symbol name so that the 2<sup>nd</sup> and 3<sup>rd</sup> characters are the FAMILY type of component i.e. **HTK1**\_RESERVOIR TANK.

The naming of symbols **must follow** AutoCAD® Electricals' naming convention.

An easy guide is:

1 <sup>st</sup> character	V or H for vertical or horizontal
2 <sup>nd</sup> /3 <sup>rd</sup> characters	Indicate the family type and can match the symbol to a catalogue lookup table
4 <sup>th</sup> character	Can be a "1" for a parent symbol, "2" for a child symbol or user-defined for other symbol types

For more information, please see **Symbol Naming Conventions** within the Help.

Normally the 2<sup>nd</sup> and 3<sup>rd</sup> digits of a symbol name govern the catalogue table that AutoCAD® Electrical looks at when assigning a part. By defining a value in the WDBLKNAM attribute, this is overwritten.

Ensure the library file path is to the correct library that you have created the symbol for:

e.g.

Hydraulics	HYD_mm
Pneumatics	PNEU_mm
Process & Instrumentation	PID_mm

Select  to review any possible errors in the symbol creation

Select

Select  to insert the symbol into the drawing