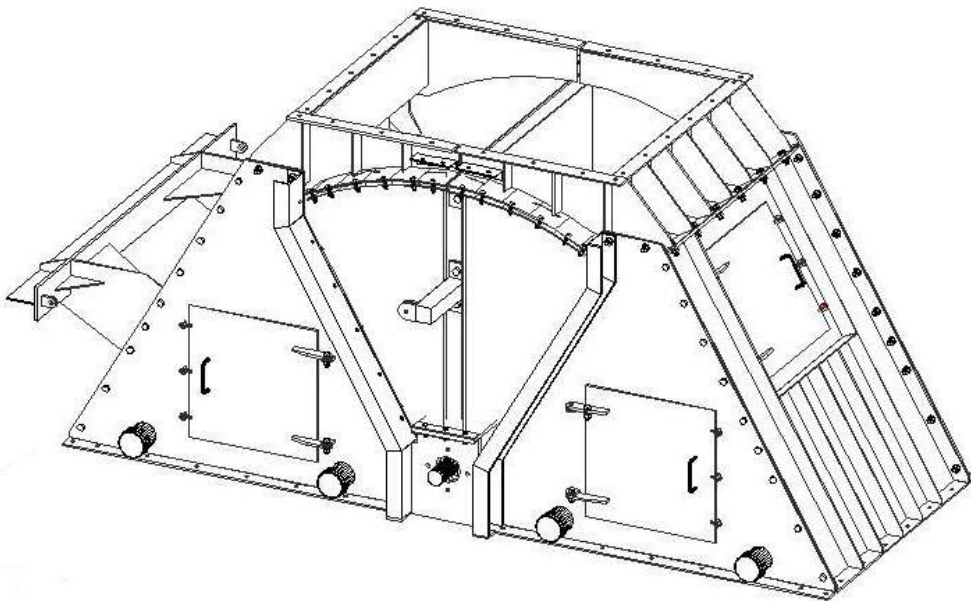


ProSteel Tutorial

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Chute model from existing fabrication drawings – Eddie Saez

ProSteel

Students should have a thorough knowledge of AutoCAD and preferably, some 3D experience.

Overview

Lesson Outcome

Students will be able to, at the end of this lesson.

- Model a 3D drawing.
- Manipulate the detail style so as to dimension in a similar way as to what they are use to.
- Create a 2D-workshop drawing, fully dimensioned.
- Create a Holding down bolt plan dimensioned.
- Create sections required.
- Create a marking plan.

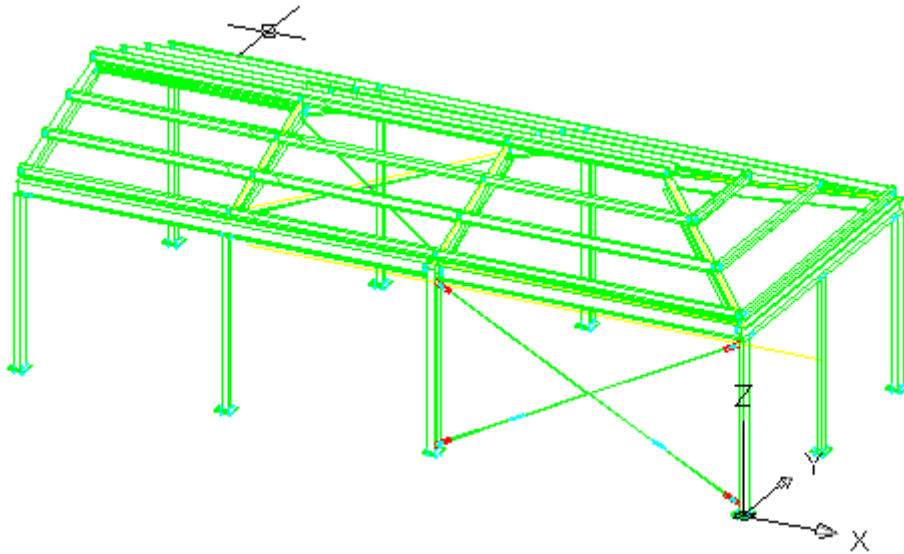


Fig 1

Work Frames

1. Draw 3D shape (Fig 2) using a mixture of Prosteel's, work frames and created PS_CONST lines, a lot of buildings do not use a rectangular or square frame so any additional lines are best kept in the PS_CONST layer for editing purposes. These lines on this layer can be erased at the press of an icon or the layer could be turned off. Use frame for overalls and grids. The work frames create views at grids, top, ends & sides etc. and these can be used for the marking plans, sections through the building and HDB Plans. Note: set view to Isometric Overview 1, as per the Aeroplane Icon.

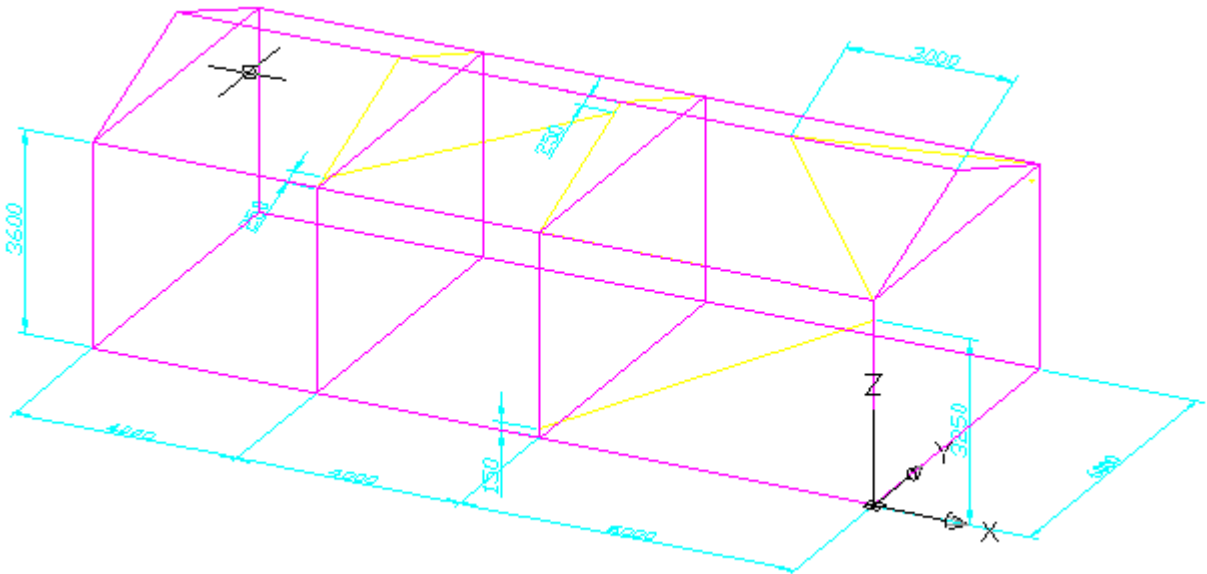


Fig 2

2. Work frames are created by selecting the work frame icon which is the top left icon on the vertical tool bar Fig 3. It will then ask you to select the work frame origin and direction of the X axis. This layout is up to you as to how you want it in your drawing or screen layout. A dialog box will appear as below and this will have options available as per the TABs ie. Rectangular, Cylindrical etc. If you move the Dialog box you will see the selected work frame in the ACAD window.
3. We will only look at the Rectangular and axis description TABs in this lesson. The other TABs are very similar and once one is understood then the others should be obvious. This Rectangular dialog box gives us 4 @ 5000 grids in the Y direction and 3 @ 5000 in the X direction and 4 @ 1666 in the height Fig 3. Note that this is a set grid pattern, however if you untick the length, width or height buttons you will notice that the axis distance buttons will change to white from being greyed out and this will enable you to have variable grid spacing Fig 4.

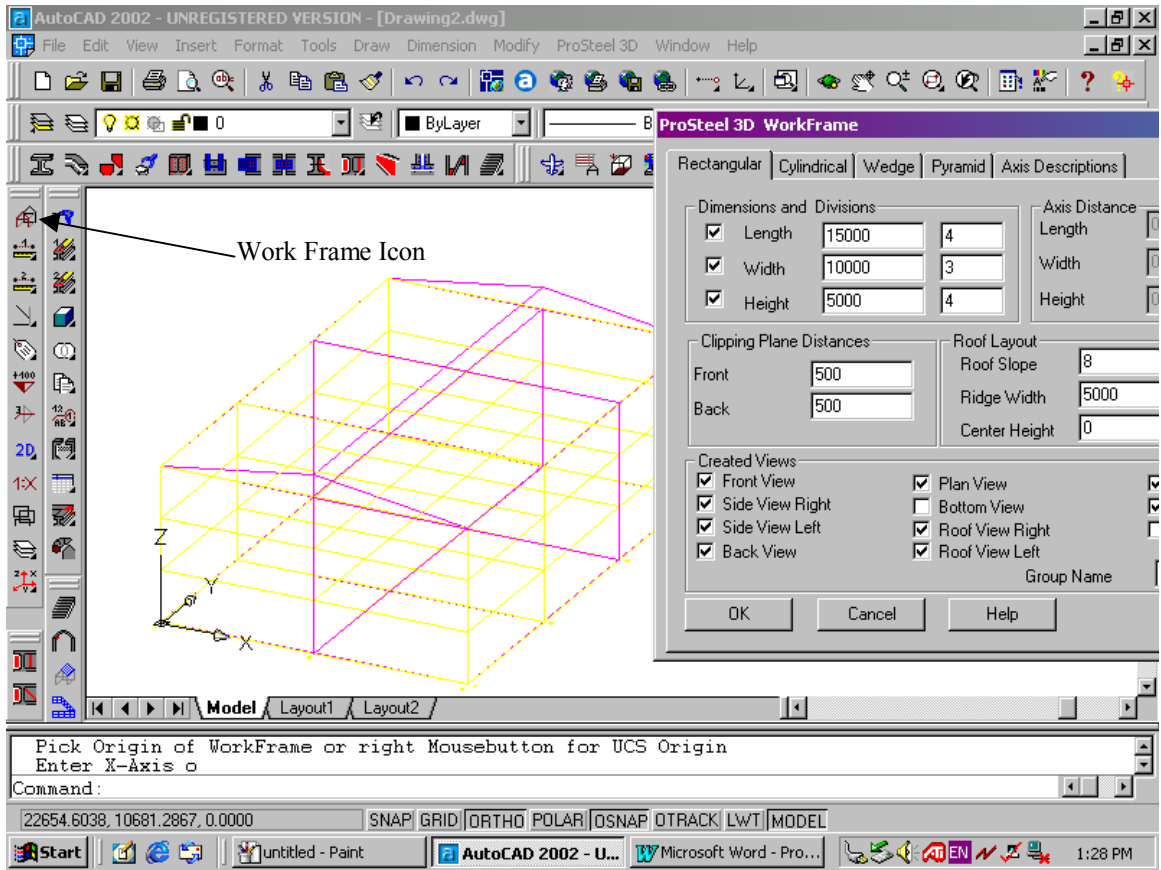


Fig 3

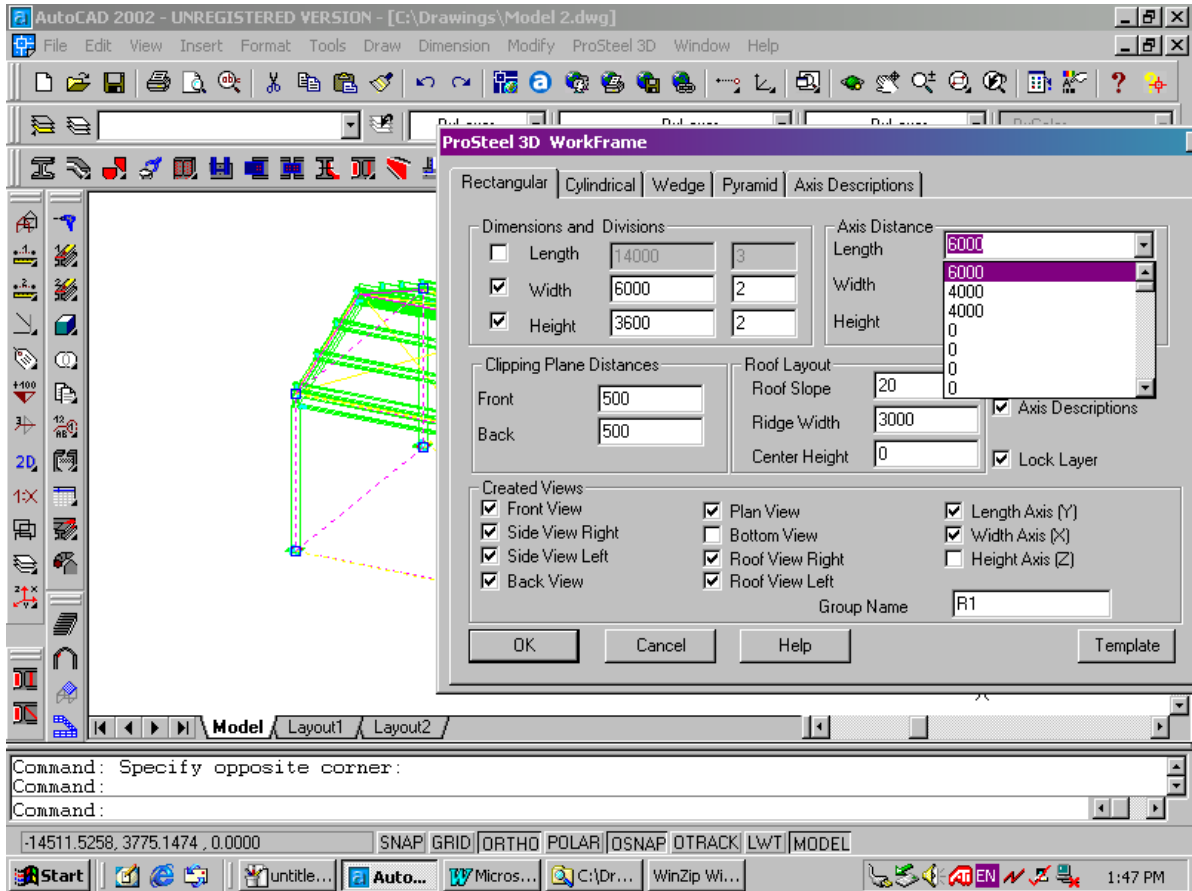


Fig 4

4. The above grid (Fig 4) set out is the way we will set up today's lesson. Note the ticked created views and the length, width & height ticked boxes. These are the views that will be created as mentioned in item 1. Note that when the 3D pattern box is ticked, which is obscured in the above Fig 4, a Yellow Grid is displayed as well as the Magenta View Planes.
5. Note the template button, this will create a copy of the set-up that you have made and you can use it at a later time on another model. This template ability is common to most ProSteel commands and allows you to set-up a library of connections, frames & plates etc. These can be opened, then edited and saved as another Template or copied over the existing one.

Adding Members to Work Frames

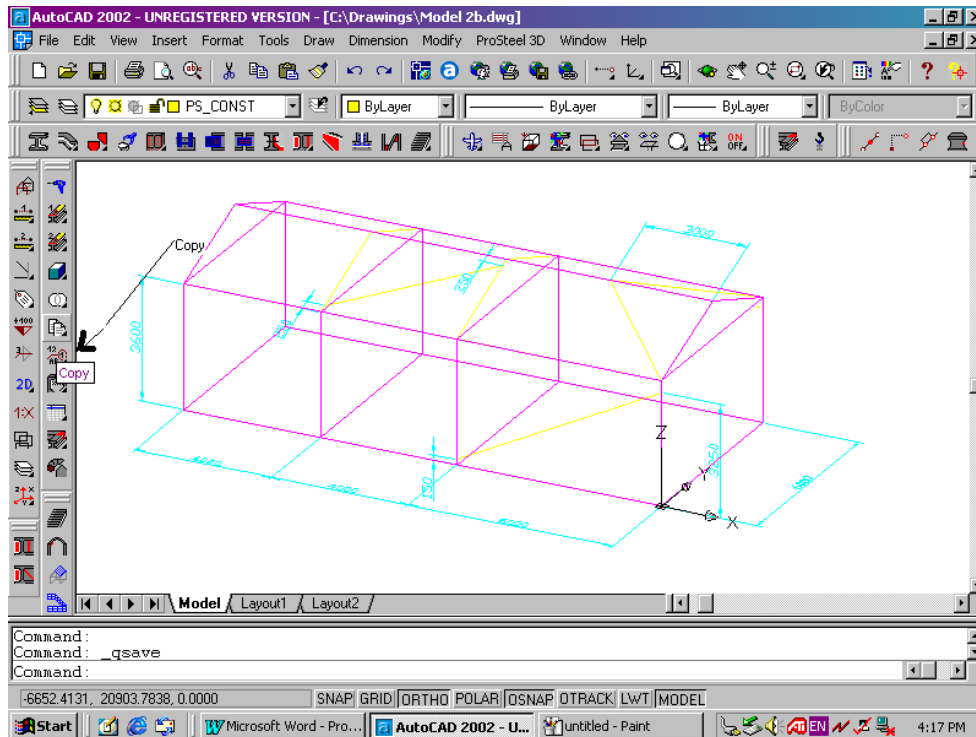


Fig 5

1. Copying frames and columns ie. "ProSteel Objects" is one way to go, however they don't show if shaded unless drawing is reopened and also the logical link is lost. So in most cases, if copying, only copy whole groups using ProSteel's copy command and make sure the logical link is still active. See Fig 5
2. If Members are mitred for instance they will maintain their link if moved or stretched, the same applies for connected members ie. shear plate beam to column.
3. After you create the work frame we need to add more lines representing the hip rafter "top centre" the intermediate rafters "top centre" and the bracing "mid point" lines on the roof and wall.
4. Draw these lines on the construction layer and use normal Autocad commands for constructing them ie. from line from end to perpendicular.
5. I have found it easier to place roof members to the mid point top of steel and columns to the centre point as per FIG 6

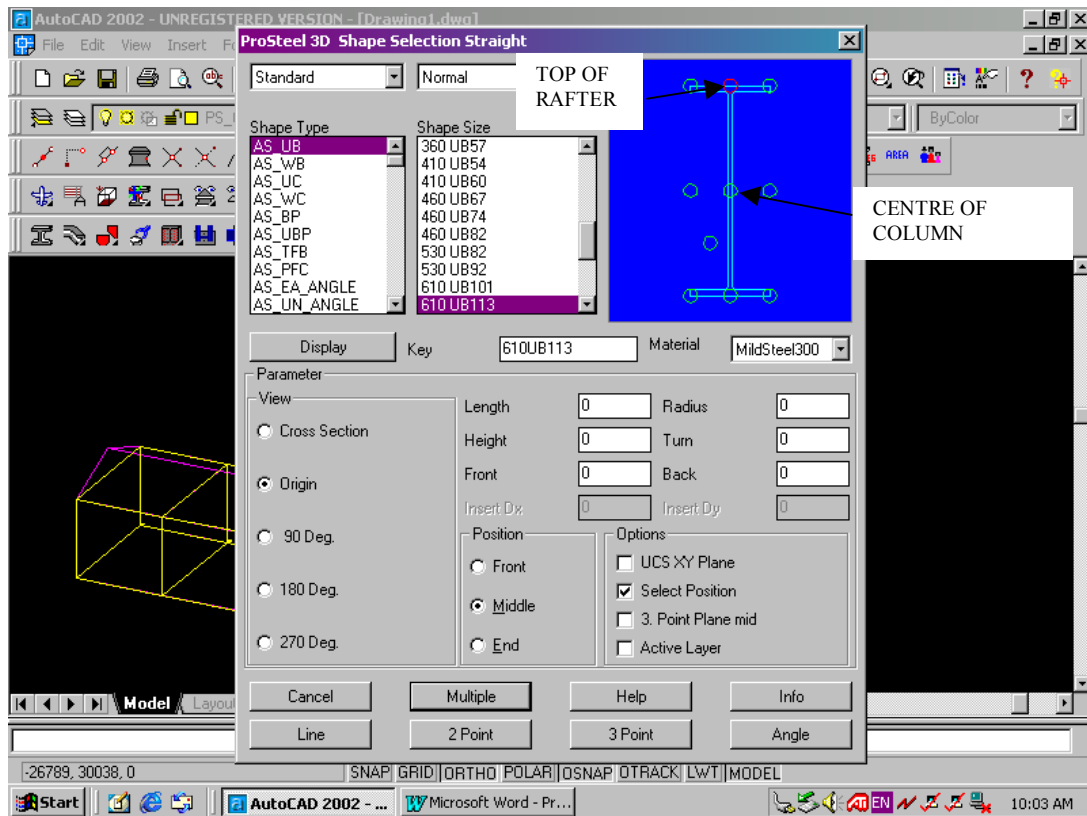


Fig 6

6. Once the wire frame basic shape has been created we can start adding members to our line. This is achieved by selecting our SHAPE icon that looks like a steel section, which is the first icon on the first tool-bar above our drawing window in Fig 6. It will then open a dialog box as per Fig 6.
7. Add members to lines by clicking on the line. You can also add by selecting two points and then click on the two points you want. You will be prompted for direction. Providing you have it set for origin as per Fig 6 then it should come in at the right orientation. Otherwise you can rotate it by clicking anywhere on the screen and it will rotate 90° per click. Press enter when ready and it will then ask for another line to add a member to. If you want to add to another, click the next line, otherwise press enter and this will finish the command.
8. Add all members required, as per the above steps.

Connecting Members

1. Connections to members. There are numerous connection that can be used and the icons reside on the same tool-bar as the SHAPE icon and they start next to the icon that looks like a bolt and spanner and run to the right. They are END, BASE & SHEAR plate, SPLICE & PURLIN. Note: some of these icons have variants on the fly-out icons.
2. We will use Shear plates, Base plate's & end plates etc and look at the options in the dialog boxes.

Shear Plate

3. We will look at the dialog box for a shear plate first as per Fig 7. The first value is the GAP value, which is the value that one beam is away from the other beams flange. The vertical offset is just what it says and the default is on centre. The tick boxes under ie. “From Edge” relates directly to this offset.
4. The weld tick box is ticked if you don’t want it bolted, the Shear Plate is already welded to it corresponding member and the “As” value is the weld size.

Bolts

5. Bolts are self explanatory with the “16” being the bolt size and the “2” being the clearance.
6. The “Form Group” should be ticked as this saves time latter adding parts to groups. Only if you wanted to separate the part to another member would you not have them grouped. The same applies for the “With Bolts”.

Cope

7. The “Cope” should only be ticked when you want to add a cope and a prototype cope needs to be selected in the scroll box below before the cope will work. This cope is added to two beams at 90° to each other through the “Manipulate Icon” and saved as a Template. This Cope Template will then appear in the Scroll Box and can be selected.

The “Cut Plate” box is ticked if you have a cleat that is not square to the web of the member that it’s attached to. See Fig 8

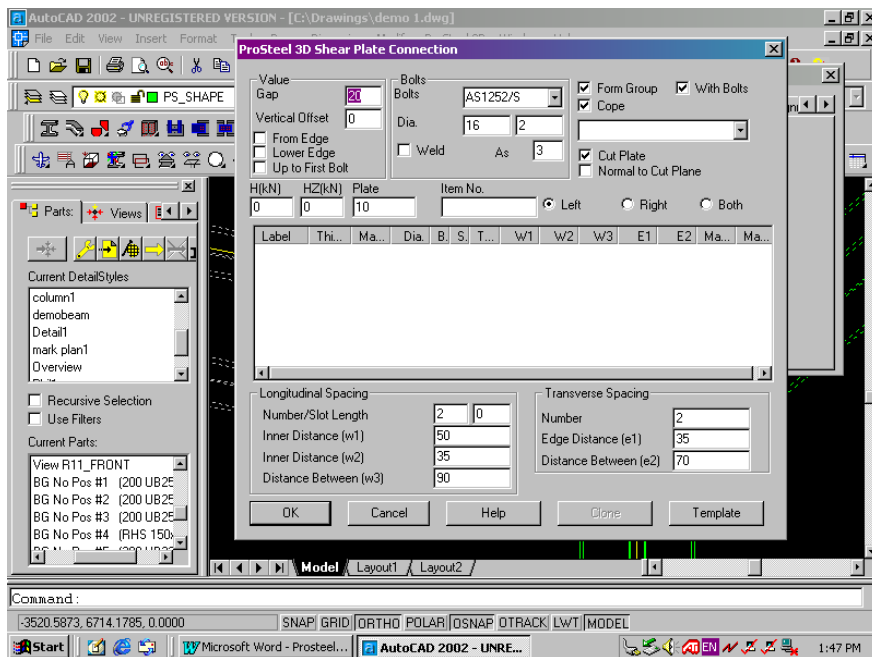


Fig 7

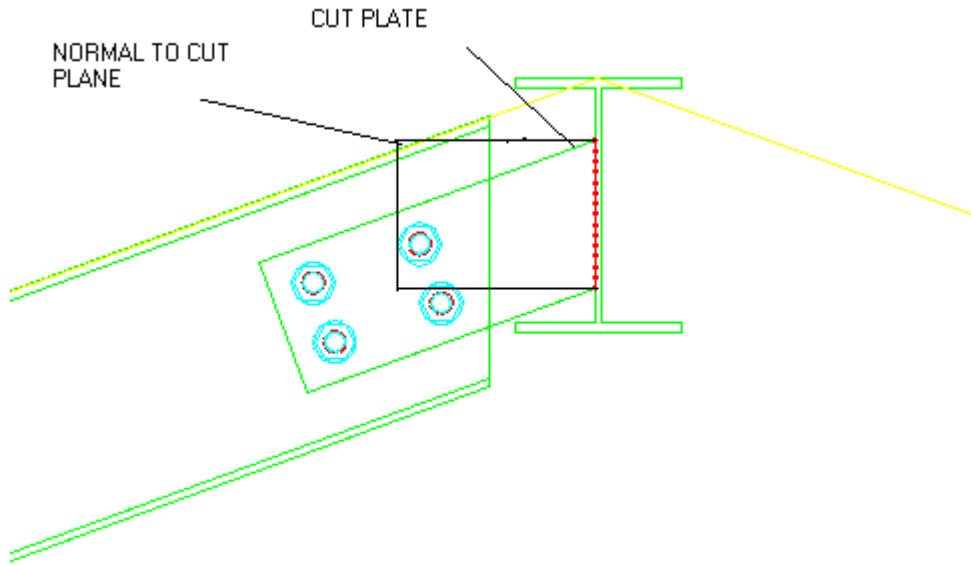


Fig 8

Normal to cut plane

The “*Normal to cut plane*” would square the cleat from the web and therefore, “*Cut Plate*” would also not be needed. Notice the red squiggly line near the web. This is the weld. If the cleat is not square to the web and if the and cut plate is not ticked then the cleat will not match the web and a rectangular cleat will be drawn.

The “*Left, Right & Both*” is the side of the web of the joining member that you want the cleat or cleats bolted to.

The table is not functioning at this stage, but will in the future allow you to specify loadings and a suitable cleat will be recommended.

The “*Longitudinal & Transverse Spacings*” determine the plate size. The “*w1, w2, w3, e1 & e2*” are distances from plate or member edges to hole centres. As all editing is real time than it is simply a matter of altering the values in these boxes and see what happens on the screen.

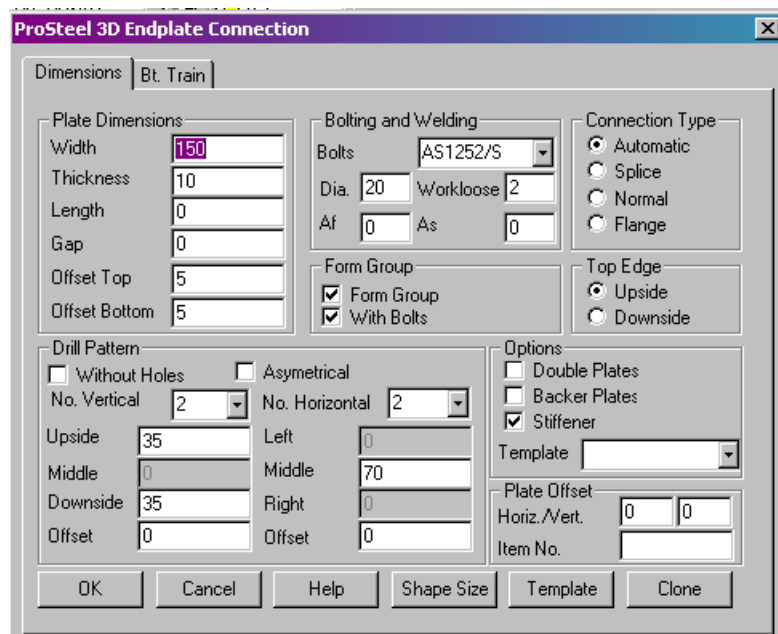


Fig 9

Endplates

8. Fig 9 is an “*Endplate*” connection and again it is easier to alter settings and see how they behave on the screen. However I will mention some items on the above dialog box. Here the holes don’t dictate the plate size and you either have to enter a width and length or enter 0 in both or one and the member default size is used. The “*Offset top & bottom*” is a value that the plate is either shortened or lengthened by (-5 to lengthen). You can have numerous bolt configurations and you can offset the plates or holes, add backing plate’s, stiffeners or add a plate to the matching members flange if it is an adjacent connection “*Double Plates*”.

Editing

1. To edit a connection, of any sort, to a member, you need to go through the links in context dialog box. This can be achieved by clicking on the member that has the connection attached to it, by this I mean the first member that was selected to make the connection ie. Column to beam, you would select the column. Then right click the mouse and select the PS3D Properties. This takes us to a dialog box, Fig 10, that has all the components of the member. Here we can edit anything relating to this member. If we select the “*LogLink*” tab and then press the “<<” or “>>” button, we will see the member highlight when we have chosen the right connection to the corresponding member. Also note the link type. If it is greyed out then you have most likely chosen the wrong member to edit. Choose the corresponding member and try again. Once it is you select the modify button and edit the original dialog box for the connection etc. Note: all editing is done this way.

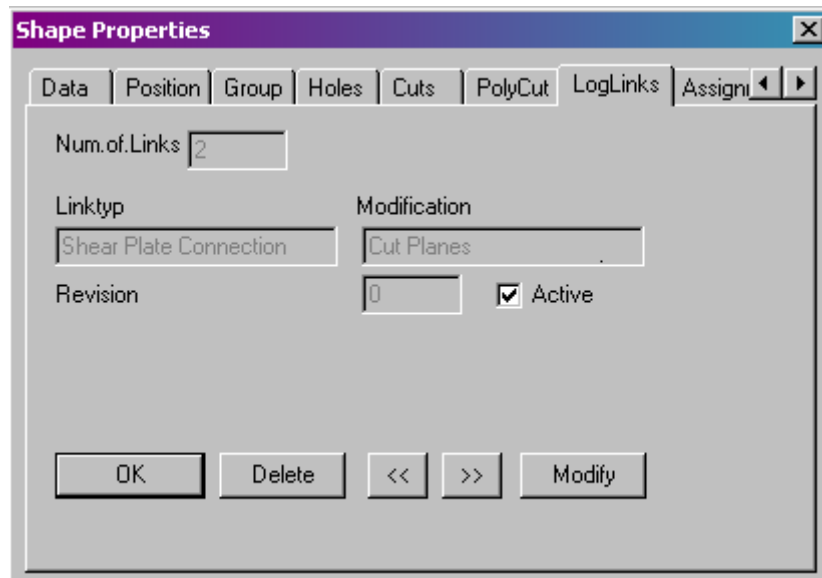


Fig 10

2. If we don’t have our “*active*” ticked then it can’t be edited and all updates to the corresponding or main member won’t be reflected in the cleat. However it can be edited by selecting the cleat and then the context menu for the cleat.
3. You can change the length of a member through the context dialog box, stretching, & grips. Also “*Modify*”.
4. You can change the section or material of members through context dialog. The offsets and rotation can also be altered. Click on the Tab you want to edit

and follow the prompts in the box and the member or connection will redraw itself as you edit it.

Grouping

1. All members must be grouped to their respective cleats, base plate's etc. Generally this is accomplished automatically if the “*form groups*” box is ticked in the individual connection boxes. Also if you want your bolts to be grouped with your members so you can create a bolt list then it is in this connection box that you choose “*With Bolts*”
2. If you added copied parts and you have not grouped some connection or parts, then for these parts to be added to the final 2D drawing then you have to group them. Consider Grouping as welding. All parts that are welded to the member have to be grouped to the member.

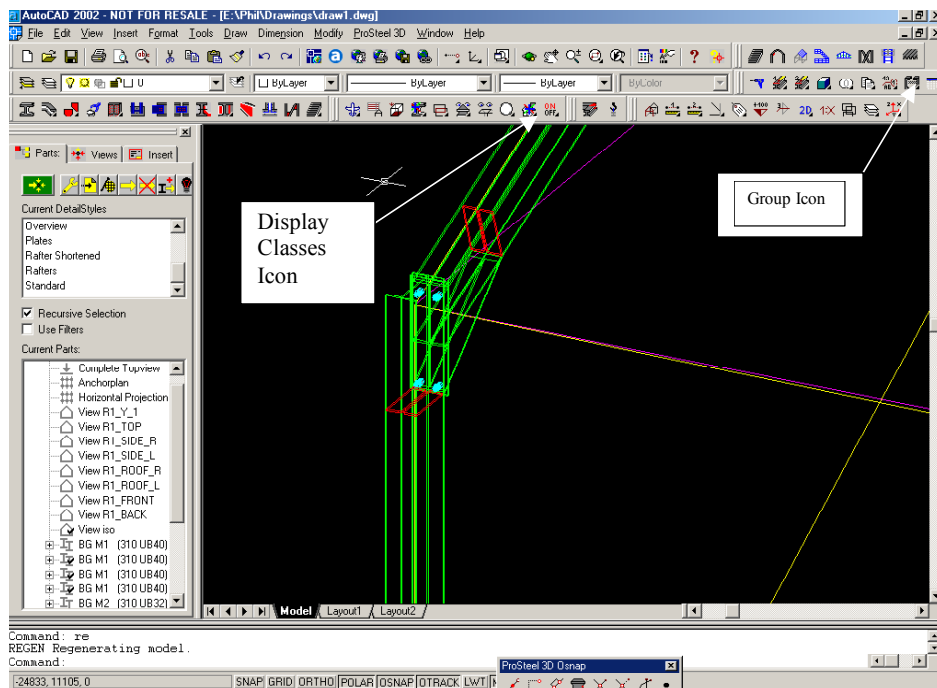


Fig 11

3. To check what is grouped then we need to click the “*check groups*” button, Fig 12, and this will show us what is grouped by vanishing all that is grouped. This will allow you to group what is left on the screen. To bring all the members and connections back again click “*Display Groups*”. If you have some connections that are missing on the members that have already been grouped, then you use “*Add to Group*”. Note: A connection member or part can only belong to one group.
4. There must be a main part when grouping items. The main part can be the member or the connection. *Tip*: It may be beneficial to create a very small user shape and make this your main part as all the alignment for views and measurement will be from the main part, this is extremely handy for trusses.

- Some members may not have been grouped at all automatically and can be grouped by selecting “*Create Group*”. This generally applies to beams with only holes and no other connections added to the member.

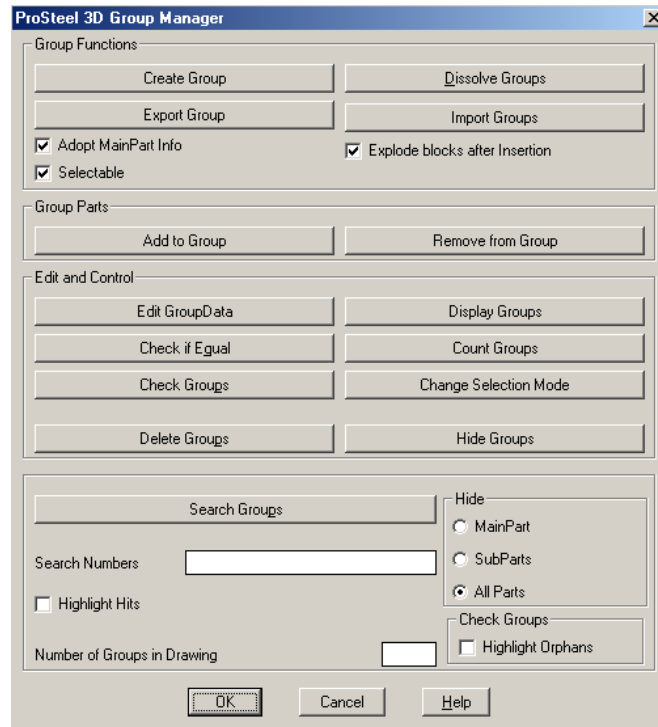


Fig 12

Display Classes

- A display class allows you to group individual groups providing “*Complete Groups*” is ticked on. The icon for display classes is the Icon that says On/Off as in Fig 11. It will allow you to put all your columns in one Class and all your rafters in another etc. You do this by double clicking in the “*class name*” area on one of the blank lines and typing in the new class name, then whilst highlighted assign the members to the class. You can then turn them off at your will by clicking and highlighting the Class and then clicking “*hide class*”. *Tip* under the same Icon on the Fly out is a yellow icon called hide except. This will hide everything except for the items chosen and hide except groups will do the same but will leave the whole group that you selected.
- There are also Area classes and Parts Families that work similar. See your Manual.

Positioning

- All members once they have been grouped will then need to be positioned, what this will do is to add a number, either numeric or alphanumeric, to the group. It will also add a data number for each individual part of the group. The icon for “*positioning*” is just left to the “*group*” icon.
- What you will have once a group has been positioned is all parts of the group will have the same group number i.e. “C1” and all parts including the main group part will have a unique part number. Please note that this is the way I set them up as it suits the applications that I teach, however there are many options available in the positioning dialog box, see Fig. 13 & 14.

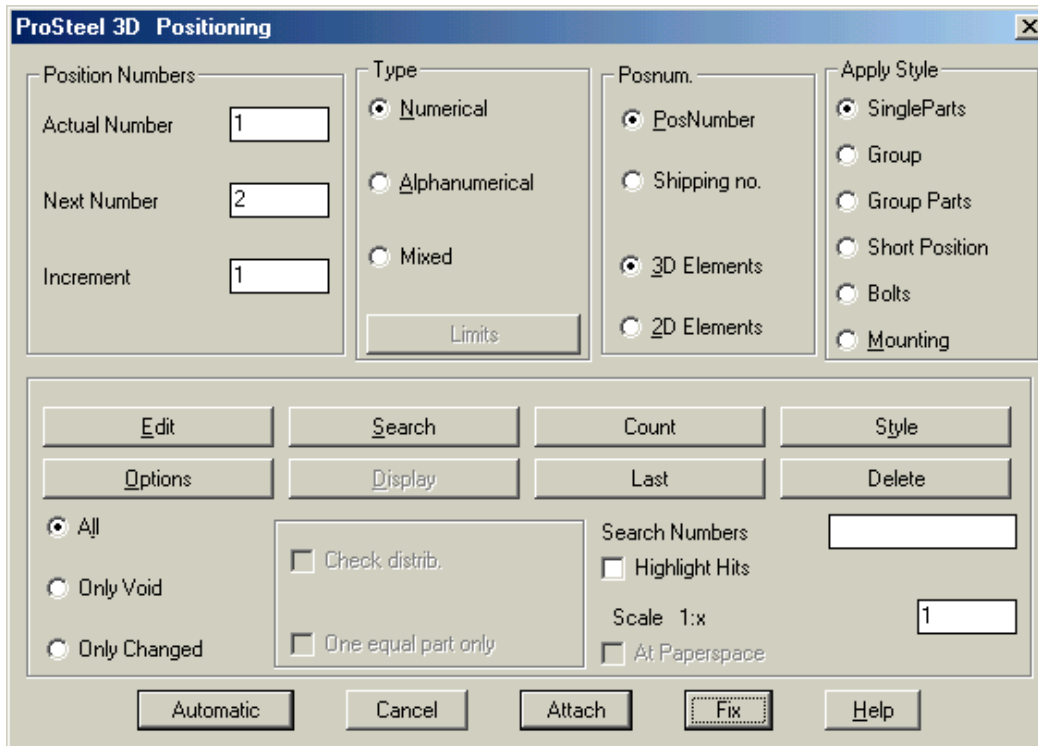


Fig. 13

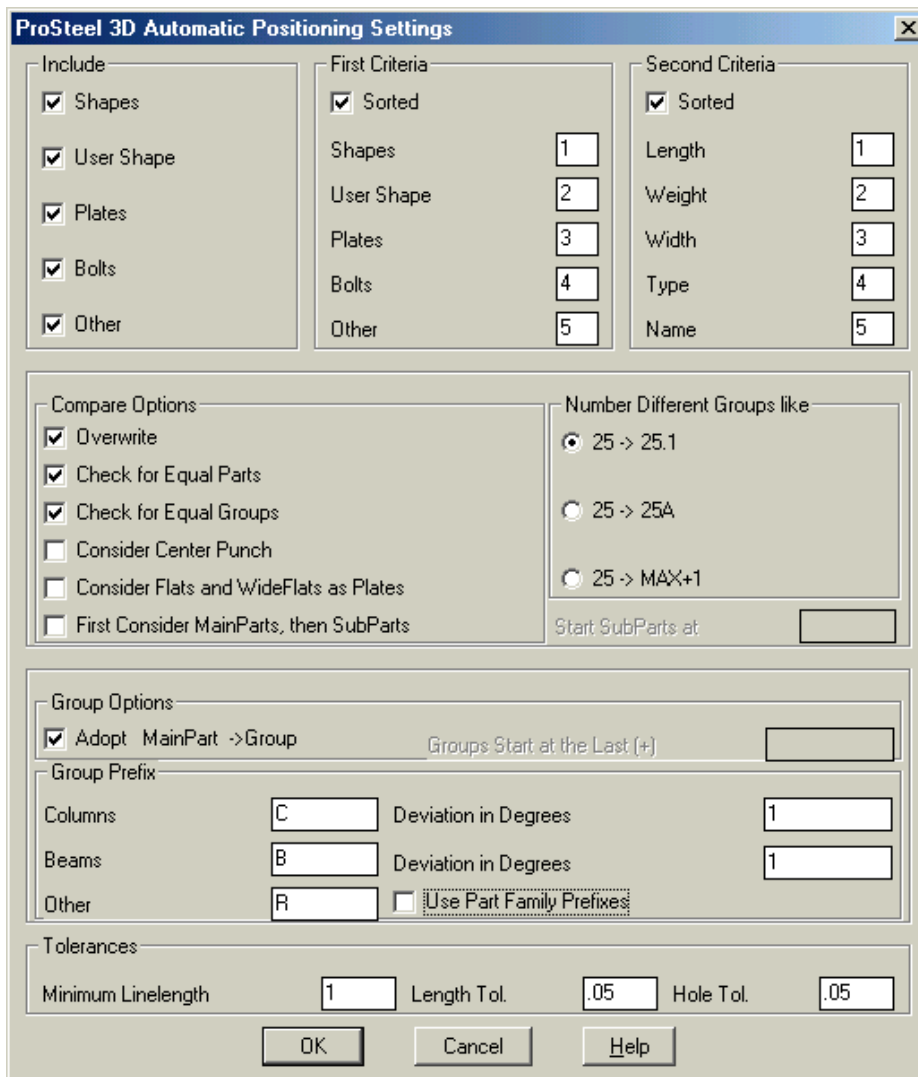


Fig. 14

3. Fig. 14 is the dialog box that is displayed when the options tag is pressed on the Fig. 13 dialog box.
4. Include means what will be given a data number so you can have all your bolts with a unique number, if you so choose.
5. Overwrite ticked means that all the numbers that had been assigned to the members, will now be reassigned a number when automatic selection is next used.
6. “*Checked for an Equal Part*” is the selection of parts and groups that have the same length and material and holes i.e. properties as another member. They will then be given the same data numbers and group numbers if this is selected.
7. “*Checked for an Equal Group*” is the selection of groups that have the above criteria but also have parts welded in the same position as the other matching group. Note the parts will need to have the same data numbers for this to work. For other options refer to the manual page 245.
8. Group prefixes. If you put a “C” for instance in the box next to Columns than this Prefix will precede the position number for all columns providing that a value of at least 1 is placed in the next box that says “*Deviation in degrees*”. This means that all members that are one degree one degree off the perpendicular will be assigned as a column. The same applies to beams however they are one degree off the horizontal. All angles will be deemed as others. “*Use Part Family*” this box is ticked so as to use the prefix that is assigned by the Part family command “PS_FAMILY_CLASS” see manual.
9. Fig. 14 is the way I set up my options box but there a many variations.

Detail Centre

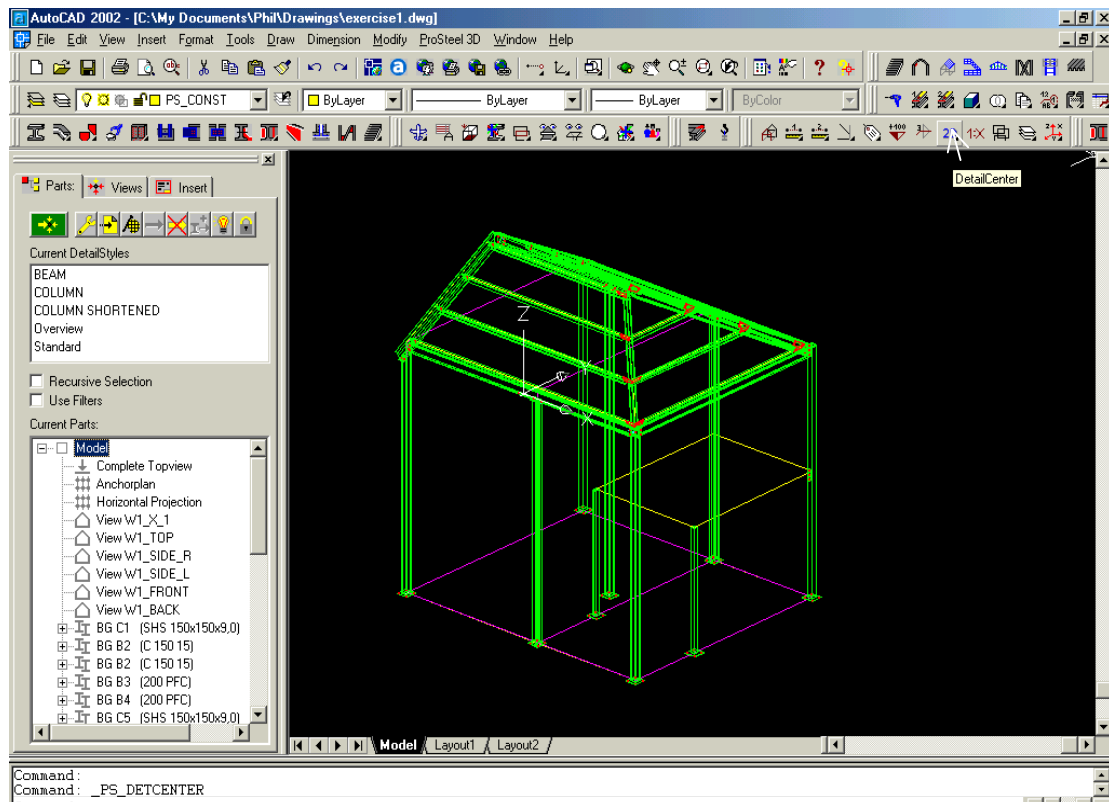


Fig. 15

1. To open the detail centre you click on the 2D button and a series of windows will open on the left side of your drawing with icon buttons attached. Fig. 16.
2. This is where you go from a model to a detailed shop drawing and it has many options but I will only explain the ones needed to get us to a finished product.

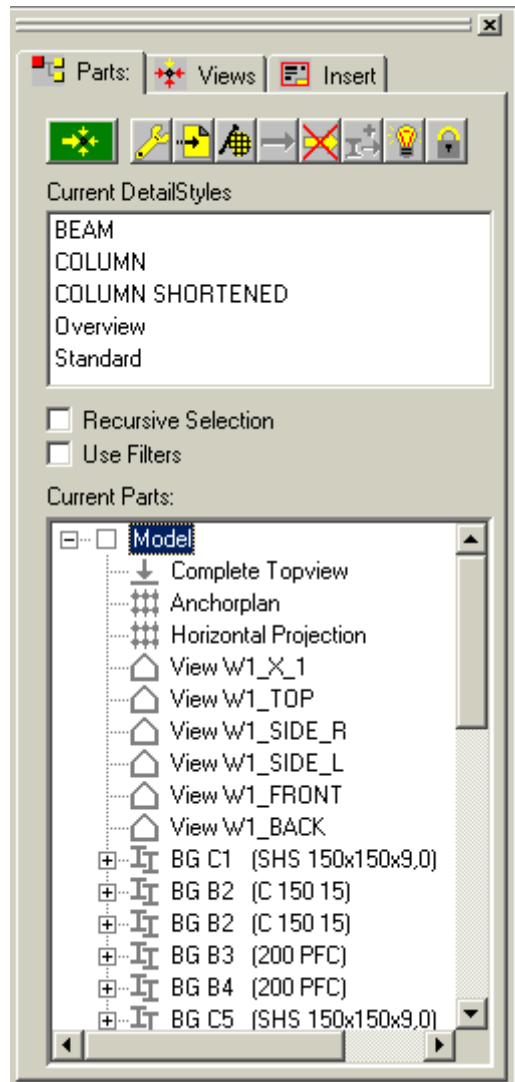


Fig. 16

3. There are two ways of using this detail centre, one is manually and the other is automatically with the “*Detail Centre Express*”

Manually

1. Fig. 16 above has nine views in “*Current Parts*” that have been created from the Work frames as mentioned in WORK FRAMES item 1. They are “*Complete Topview*” down to “*View W1_BACK*”. The rest of the items in that column refer to the individual members that have been grouped. Above in the “*Current DetailStyles*” are the detail styles called “*BEAM*” “*COLUMN*” etc.
2. I suggest that you load the detail styles that come with the system and edit those detail styles to suit your own applications. To do this you click on the “*Standard*” detail style and using your right hand button on the mouse choose

to load all detail styles. This will load all the detail styles that come with your system.

3. Once they are loaded, double click on the style that you want to edit. The style will now look like Fig. 17 with the name of the style up the top. I have chosen Standard. The first to edit is to change the selection from DIN to American. This will change to drawing views from 1st Angle projection to 3rd Angle projection. This will need to be completed for all styles. Another thing to be carried out is to tick “*Update Preview if Detail Style Changes*”

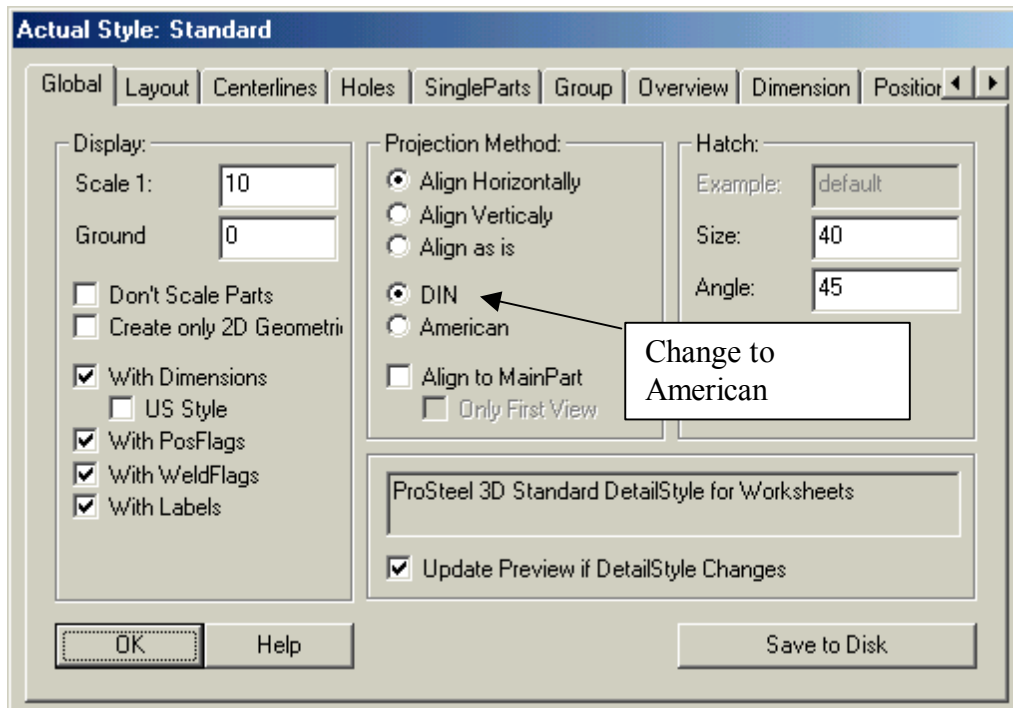


Fig. 17

4. I will not discuss the detail styles in depth as there are too many options to discuss in this training brief, however I will discuss a few options that may not be obvious. “*Don’t Scale Parts*” for instance, if ticked will place all parts on the 2D drawing full size so you will have to scale the border and plot size to print, otherwise all parts will be scaled down by the value in the “*Scale 1:*” window. By having the “*Don’t Scale Parts*” box ticked, than as I said the parts won’t be scaled however the text sizes etc. On the drawing will be scaled by the values in the “*Scale 1:*” window. Leave the “*Create only 2D Geometry* box empty. A few other options are discussed in the Edit section, Item 6 through to item 12
5. Once you are happy with your detail style you must save it to disk with the “*Save to Disk*” button.

Edit

6. The best way to edit a detail style is to first drag a current part from the “*Current Parts*” window onto the detail style in the “*Current DetailStyles*” window. Once this has been completed click the “*Views*” TAB and the part should show up in the window “*Current Views*”. See Fig. 18
7. By highlighting the part that you have the detail style attached to, you can then click your right hand mouse button to pick and highlight “*Preview*”. This will

bring up a preview of the part with the detail style that you chose to attach the part to; this detail style will determine the presentation of the part.

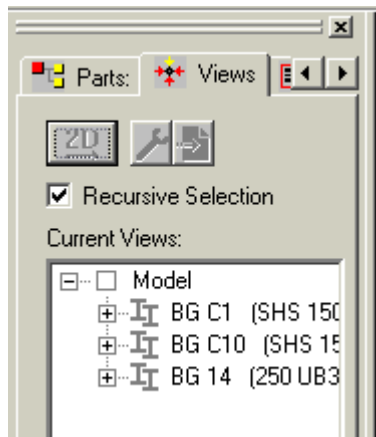


Fig. 18

- To edit this detail style and then save it to disk you must again click the member in the “*Current View*” list and with the right mouse button select “*Modify DetailStyle*”, what will happen, provided “*Update Preview if Detail Style Changes*” is ticked, is that any changes that you make to your detail style will be reflected in your preview so you can see exactly what you want and once you have what you want then save it. Note that if you don’t save it to disk the change that you make will affect the current drawing and future drawings but when you reload that detail style the old style will take effect.

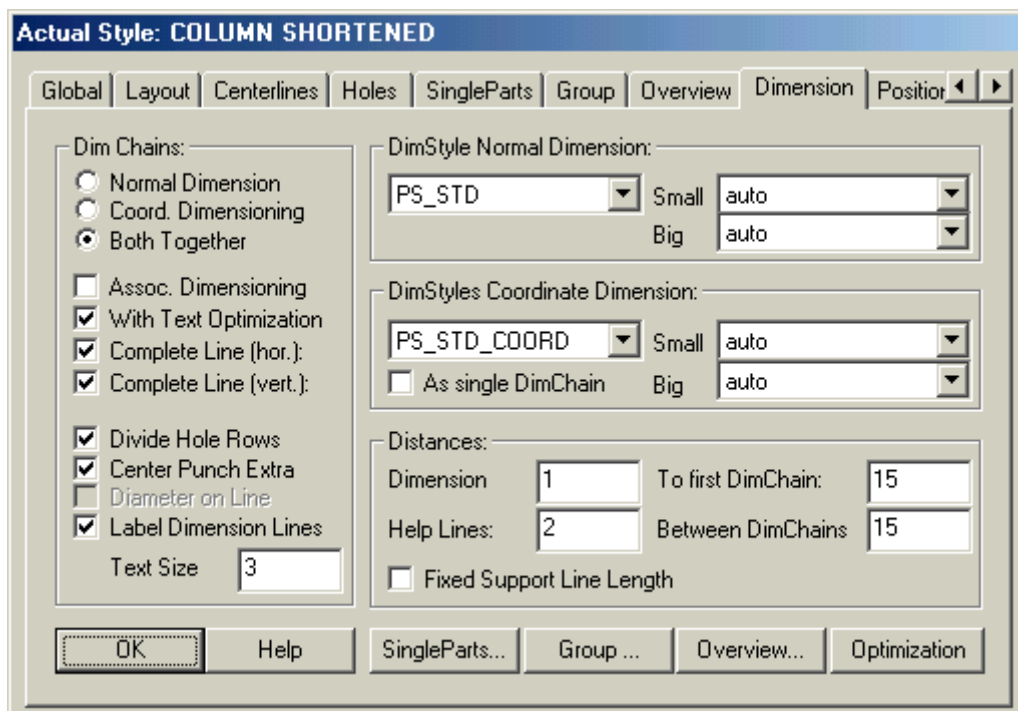


Fig. 19

- The dialog box above, opened on the TAB “*Dimension*” give us a picture of the two AutoCAD dimension style that the system uses, they are PS_STD and PS_STD_COORD. What happens is that when 2D drawings are produced ProSteel copies these AutoCAD dimension styles and renames them with an updated scale. You can edit these AutoCAD dimension styles but you must do

that in the ProSteel metric template and save it, prior to any model being created.

10. Another TAB down the bottom of Fig. 19 is the “*Overview*” button. This Overview affects the combined drawings such as marking plans and Holding Down Plans i.e. numerous groups joined together and views formed from these groups.
11. Fig. 17 above shows us that the members will be placed horizontally on the drawing. You could place them vertically for columns or Align as is for rafters, this will place the details the same angle rotations as they are in the model from the base. You could leave them all as “*Align as is*” if you wanted to and they would come in as per the model i.e. columns vertical and beams horizontal. You would probably want your sub parts i.e. base plates cleats etc to be horizontal therefore they would have to be placed on a separate detail style. This can be done easily in the “*Detail Centre Express*” but also can be done manually.

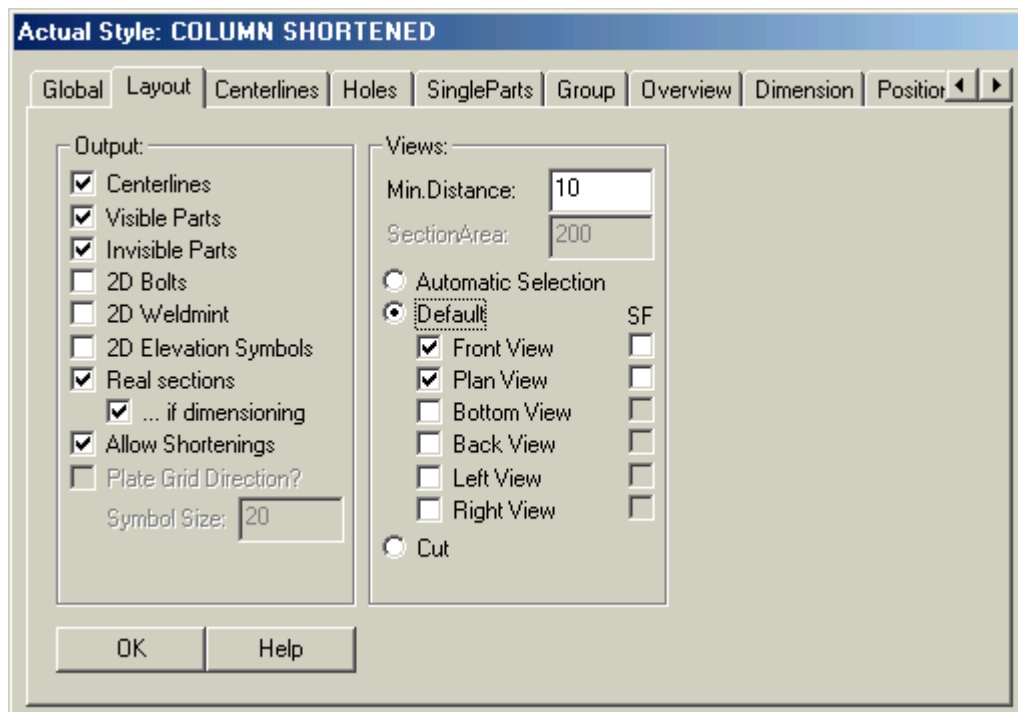


Fig. 20

12. You can choose the views that you want generated and whether you want it shortened etc.
13. Now to continue with producing the 2D drawing once you are happy with the detail style that you choose.
14. To carry on with the process you must first close your Preview. Then you drag your part from “*Current Views*” onto the 2D button. See Fig. 18. You should see it creating the views by a series of numbers flashing on the screen next to the button. Don’t worry if you can’t see them.

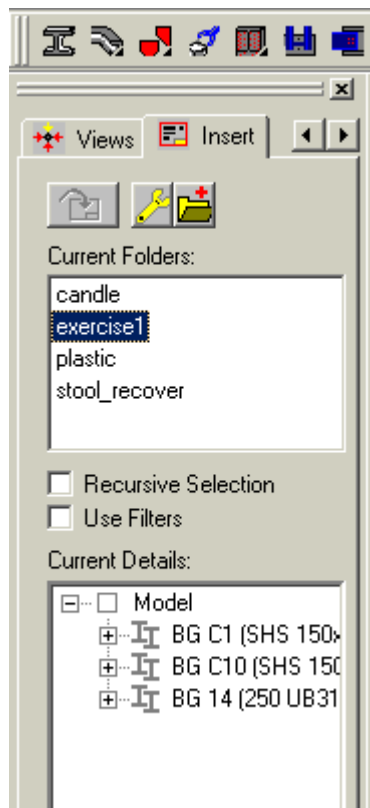


Fig. 21

15. Next click on the insert TAB and the window should change to the same as in Fig. 21. If you then click on the model name that you are using, that should now reside in the “*Current Folders:*” similar to exercise1 above, all parts that you have attached detail styles and views to will be listed in “*Current Details*”
16. The model must be saved as a name and if it had not by been saved by now than the program would have told you to save it as it cannot maintain a link to the detailed drawings if it has no paths or names to create those links.
17. Next you must open a new drawing and make sure you use the ProSteel metric template. Note the template must also be used when creating the model. Once you have this new drawing on the screen you add your border and the best way to do this is through “*Insert Drawing Border*” this Icon resides below the 2D icon on the fly out. Note: if you have trouble finding Icons then I suggest you load the whole icon bar instead as per Fig. 22 until you get use to where they reside.



Fig. 22

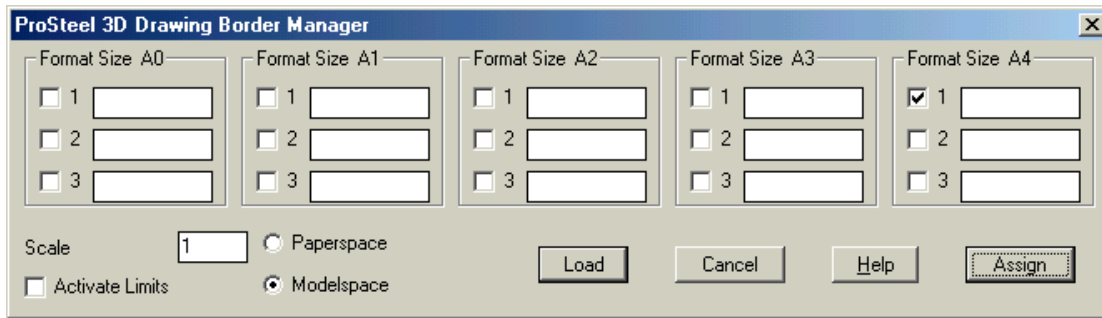


Fig. 23

18. Once the border icon has been activated you have to assign a border drawing to the Border Manager. You do this by clicking on the 1 under the size sheet you want and giving it a name next to that 1, then click the assign button

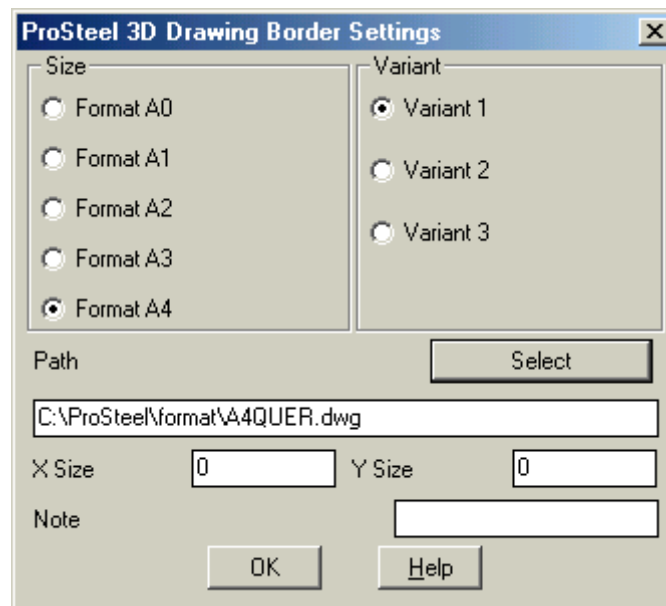


Fig. 24

19. You then select Size, Variant and select the path pressing OK when finished.
20. Next select the scale that you want as per Fig. 23 and press load.
21. Save the drawing now the border has been inserted.
22. Next drag the member that you want in the drawing from the “*Current Detail*” window onto the drawing Icon button. Fig. 21 Then as you move your mouse to the right the member block should appear in the drawing frame or border. Click on the drawing where you want this block to be deposited.
23. If you have the function “CTRL A” working then you should be able to toggle between having this as a block to all data being individual. This allows you to erase and move data easily and then block it all back together. If yours does not work then you will most likely have to edit the ACAD.MNU file and reload it. If you don’t have this function then editing is difficult as there will always be either one block for each member or the whole drawing will be a block depending on the toggle of “CTRL A”. You will need to talk to me about editing this .mnu file.

24. *Recursive Processing*: Any processing applied to a parent entry in the component parts list (e.g., assign style) is also applied to all subordinate entries. Otherwise, only this entry is processed or all equal entries, if this option is activated in the *model*. In other words a single detail style will be applied to the assembled drawing and all parts if recursive is selected. If you want the parts i.e. base plate to be attached to a separate detail style so as to show a plan only of the plate, for instance, then it would need to be placed on a separate detail style.
25. If you expand the tree Fig. 25 in the “*Current Detail*” window then you can get to all the individual parts. This also applies to the windows “*Current View*” and “*Current Parts*” on the other TAB’s. So if you where in the “*Current Parts*” window of the first TAB then you could drag a base plate or cleat plate into a different detail style if “*Recursive*” is not ticked.

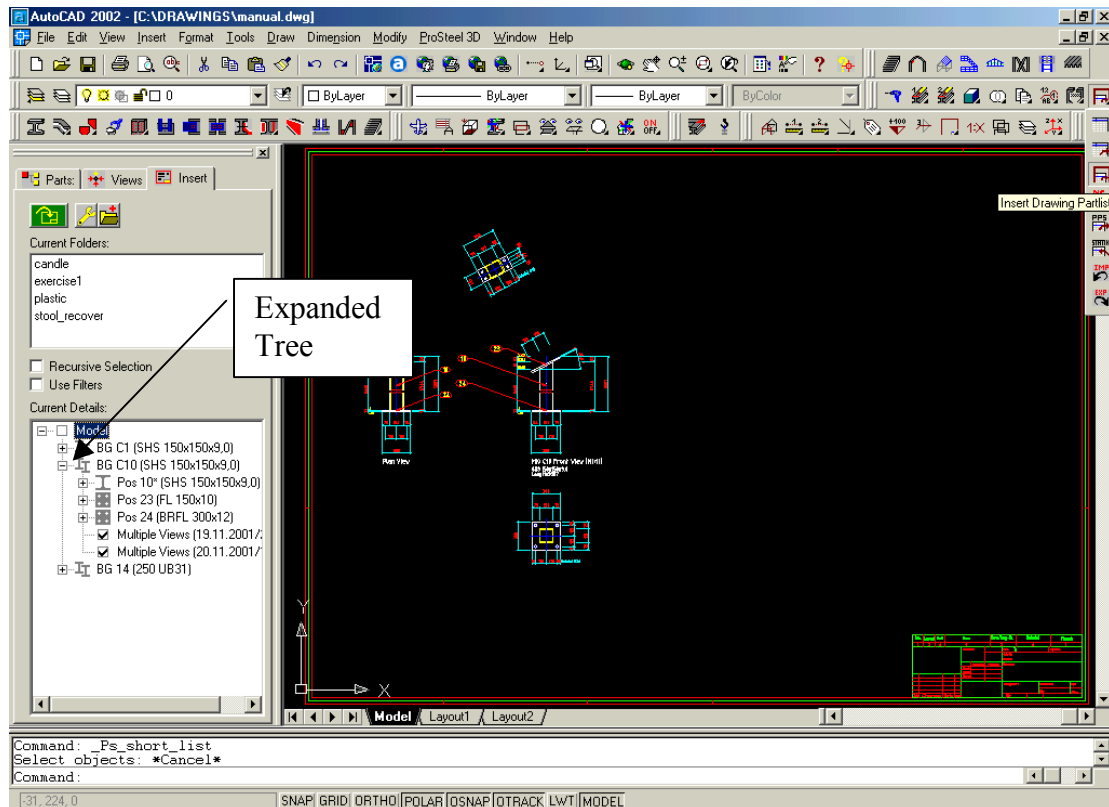


Fig. 25

26. Once all the members have been deposited then it is time to add a material list to the drawing. This can be achieved by selecting the Icon “*Select Drawing Partlist*” Fig. 25 and then a dialog box will appear Fig. 26.

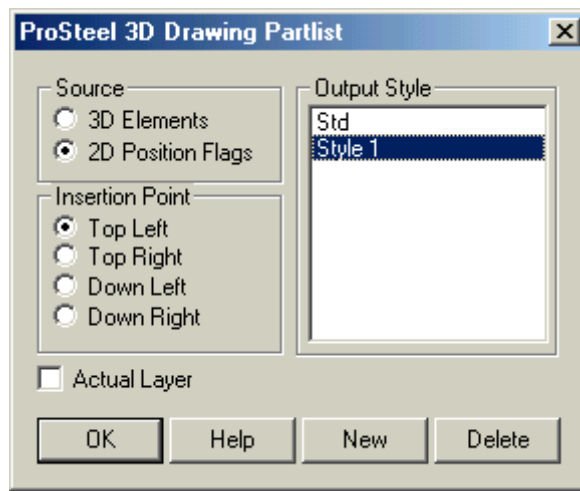


Fig. 26

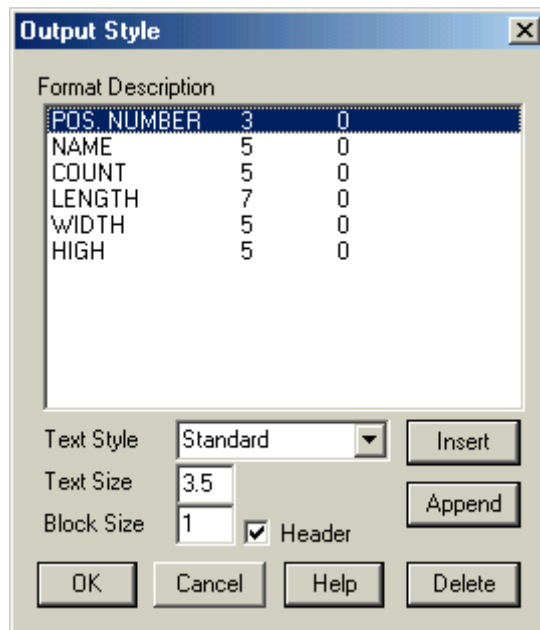


Fig. 27

27. If you go to the “New” Button (Fig. 26) you then give it a name and set up the fields that you want by double clicking on the empty line then a box comes up as per Fig. 28

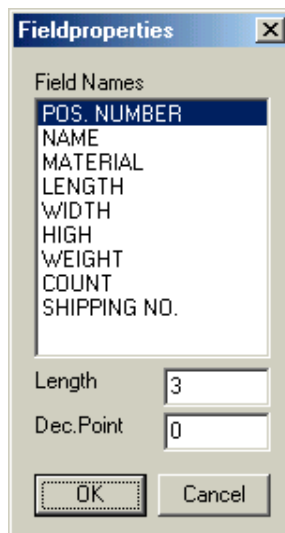


Fig. 28

28. Once this is set up you click the flags on the model that you want in the material list and press enter when finished then place the material list where you want it. Again you want CTRL A to be able to toggle so you can choose individual flags. Note: if there is more than one part with the same flag number than you will have to choose every flag to get the number count correct.
29. The 2D drawing is now complete and you can plot it out.

Automatic

1. The Detail Express Centre allows you to add detail styles to the individual parts on members as well as the members themselves. Once you process the centre it will create all the blocks for all the members that are present on the Model drawing providing that you chose to “*Update*” it first
2. This cuts out all the steps up till Item 17 in the Manual process and makes them Automatic. It will cut out all the steps and become fully automatic and place one detail per drawing including a full material list if you choose that option in the centre Fig. 30. However all the subpart drawings will be on separate drawings. You can choose the sheet size or let the computer choose it for you.

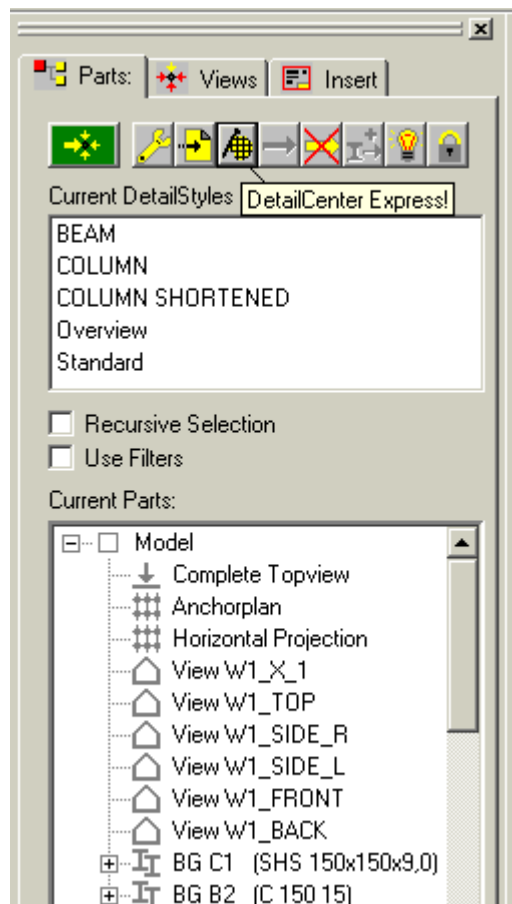


Fig. 29

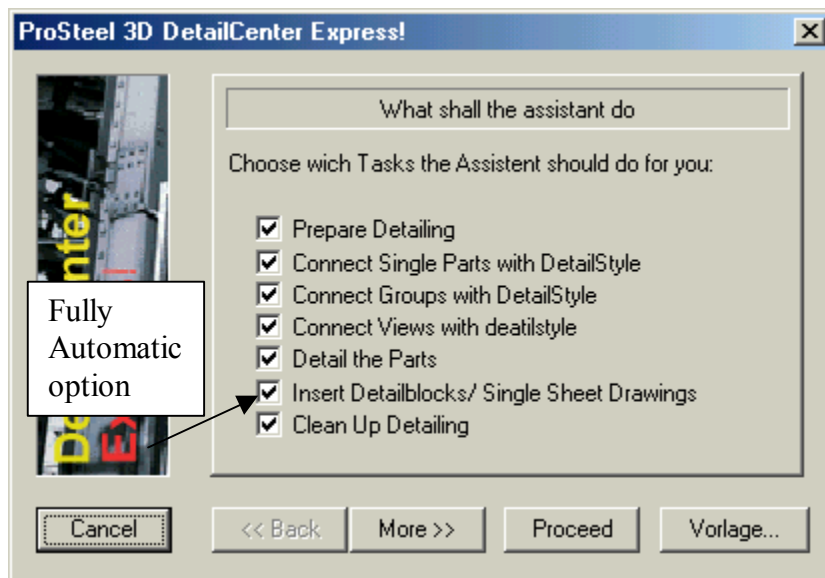


Fig. 30

3. To use this centre just tick the options that you want and go to “More”, filling out the options as you go. Once you get to the “*DesignStyle for SingleParts*” Fig. 31, place the styles that you want for your individual part. ProSteel comes with Detail Styles that should suit these selections and are named similar. However as discussed earlier they will need to be edited and saved. The same will apply with the rest of the Centre.

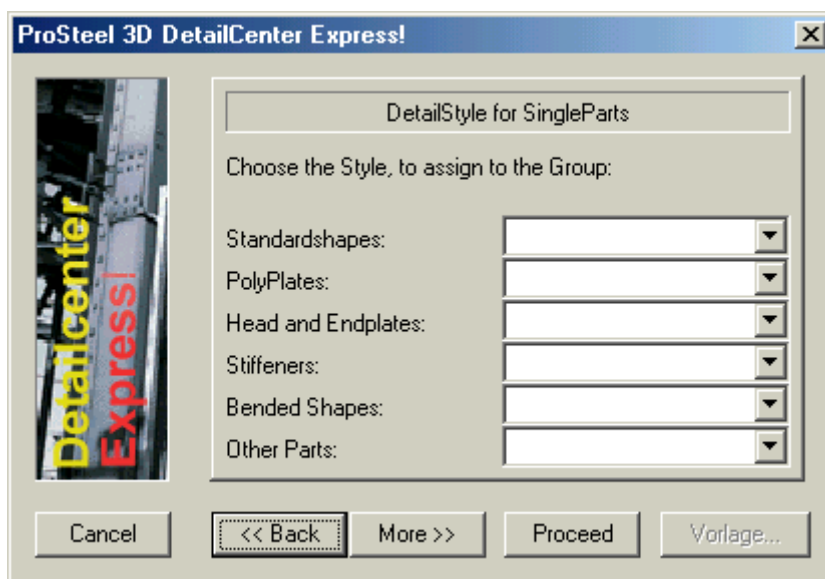


Fig.31

4. Once the centre is filled out press “*Proceed*” and all blocks or drawings will be created. This may take a little time depending on your computer.

Marking and Holding Down Plans

1. The views are already created for the Marking and Holding down bolt plan and the Detail Express Centre should have produced blocks for insertion into the final drawing. If this has to be done manually just treat the view the same as a member and drag and drop to the respective details and buttons. The named views should be “*Complete Topview*” and “*Anchorplan*” There is a detail style called “*Base Plan*” and this should be set up for the “*Anchorplan*”.

Remember that the Overview button on the Dimension TAB will affect the measurements relating to the Holding Down Plan.

2. For the “*Complete Topview*” detail style, I suggest you make a copy of the Overview Detail style and call it Marking Plan. You can do this by clicking on the detail style that you want to copy and with your right mouse button select copy then give it the name and description that you want. They can both be the same text if you want. The description can be explanatory and will appear on the global area of the detail style giving a description of the style.
3. Once you have accomplished this it may be necessary to edit this new style. The Global TAB should already have been changed to American. The TAB named Overview Fig. 32 needs to be filled out like shown below in Fig. 32. I would suggest that you edit this through a preview of the “*Complete Topview*” and then change some of the buttons and see how they alter the preview. Save the Detail style when you are happy with it.

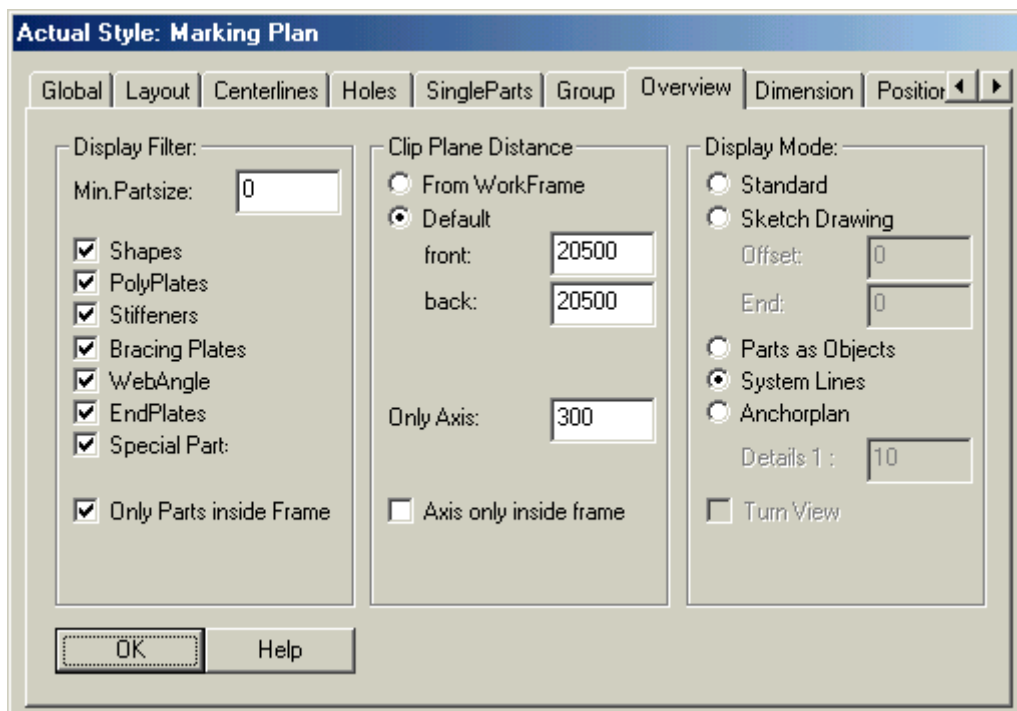


Fig. 32

4. What you have just applied to the Marking and holding down plans can be applied to sections through the building and side views etc. You can even create your own views.

Creating Views

1. Some views are automatically created when you use “*Workframes*”, but you may require more or you may not want to use “*Workframes*”. To do this you have to first set up your UCS. Say you want an Isometric drawing similar to Fig. 15 but with all flags on the members and in a 2D format. To do this you first need to set the UCS to the View by typing UCS on the command line and entering than type V and enter. This will set the UCS to the current view.

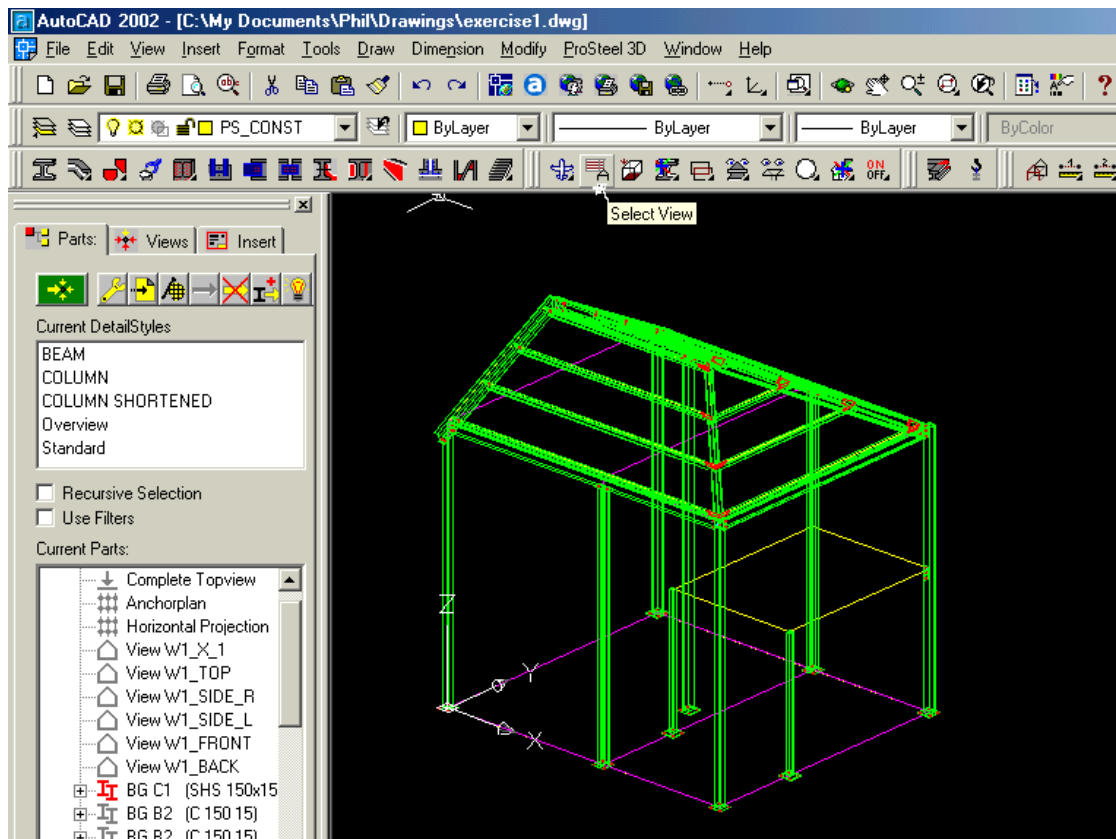


Fig. 33

- Next hit select view (Fig. 33) and a Dialog box will come up (Fig. 34) type in a name and press enter. The create button will highlight and then select this button. Press cancel once the view has been formed and is in the list.

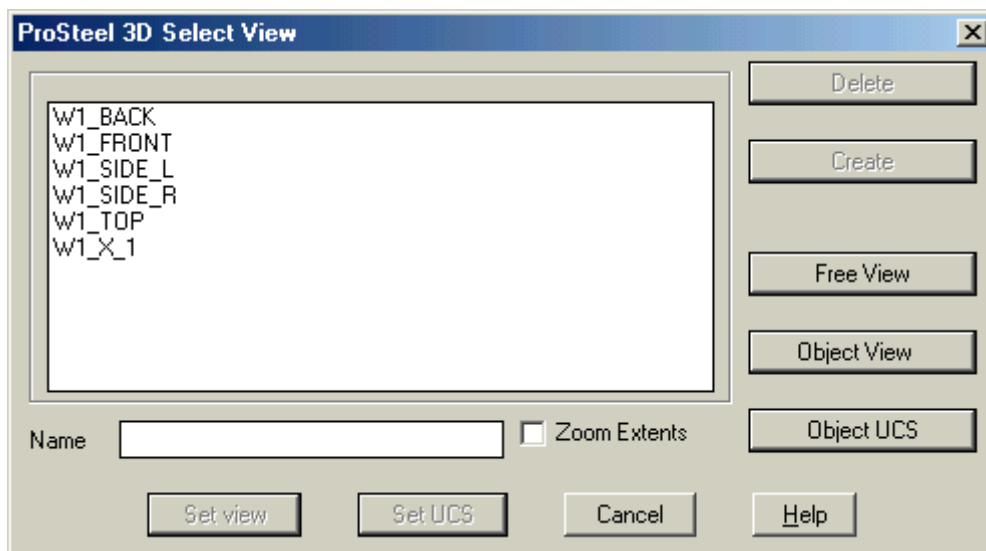


Fig. 34

- Update the model view and that view will now appear in the “*Current Parts*” window. Then drag into the detail style that you choose and repeat the procedure to get it onto a 2D drawing.

Updating an Existing drawing

- Under construction.

2. Add a material list to plan.
3. Edit using preview the detail styles and adjust the flags to be only group setting.
4. Use paper space so you can add isometric views etc. Up to individual.
5. Change the model move or shorten a beam update beam and all views.