SOLDRAW command

The **SOLDRAW** command is used to generate profile and section in layout viewport.

Command Access:

Menu : Draw > Modeling> Setup> Drawing
Command : SOLDRAW

Command Prompts:

Regenerating model.
Select viewports to draw.. Select object:

Function Description:

The SOLDRAW command could only be used in viewport created by the SOLVIEW command. Users could use the SOLDRAW command to show solid profiles and visible edges and hide lines in new viewport, and then project them onto the plane that perpendicular of observation direction. The slice plane will create profiles and edges for solids. According to section view, users could create section hatch by setting current value of the HPNAME, HPSCALE, HPANG system variables.
The profile drawings and sections in selected viewport will be removed; it will automatically regenerate new profile drawing and sections. All layers will be frozen except layers for displaying profiles and sections.
Warning : Please permanent place drawings on the layer named by "-VIS", "-HID", "-HAT". The information of those drawings will be changed when running the SOLDRAW command. Users could quit viewport created by the SOLDRAW command by UNDO command.

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