



# ACE Structural Engineering Applications LLC

## ACE FrameWorks Utilities

### Polygon Slab, Plate & Grating on Beams Documentation

Mar 15, 2013

#### Polygon Slab Plate & Grating on Beams (ACE\_PSG.MA)

(Versions: FWP 3.1.x.x/3.2.x.x rel 2.0.4 & FWP 7.0.x.x rel 7.0.4 & FWP 7.1/7.2/7.3 rel 6.0.4 & FWP 8.0.x.x rel 8.0.4 & FWP 9.0.x.x rel 9.0.4 & FWP 10.0.x.x rel 10.0.4 & FWP 11.0.x.x rel 11.0.4 & FWP 12.0.x.x rel 12.0.4)

The *Polygon Slab, Plate & Grating on Beams* application simplifies the placement of a polygon surface with from 3 to 30 sides. The polygon surface may be a concrete slab, steel plate or steel grating represented by a FrameWorks “slab element”. The utility allows as few as 3 and as many as 30 beams (any non vertical FWP member) and/or FWP grids and/or MicroStation lines (type 3 lines not linestrings) to be selected to define a polygon boundary for the surface. For FWP members w/ CP=8, the surface extent (inset or outset from the beam centerline/CP line) may be to the inside or outside edge of the steel beam flange. For all beams/grids/lines, the surface extents may be at the CP line or a defined distance from it. A positive distance or outside edge will cause the slab surface area to increase. The beams/grids/lines (termed “elements”) may be horizontal (flat) or sloped but may not be vertical. The elements must be picked in successive order. The elements are projected to the control elevation and every element must intersect the neighboring element(s). The first element defines the Control Elevation, however the elevation may be changed at any time by keyin or “last data point”. The polygon is formed by selecting intersecting elements (either clockwise or counterclockwise) successively until a closed polygon is formed. As each new element is selected, the intersection point (a polygon node) and any formed polygon sides are displayed temporarily in the view. When an option to close the polygon exists, a dialog close option will appear on the accept element dialog box. The polygon is formed by toggling the close polygon box and accepting. The polygon formation process is described in Basic Operation section of this document. The slab may be offset from the Control Elevation.

The screenshot shows the 'ACE Steel FPL Utilities' dialog box, titled 'ACESEA(c) 1998-2010'. The main section is 'Polygon Slab/Plate/Grating on Steel Beams (Model Units Feet - Input Units inch)'. It contains two main sections: 'Polygon Slab/Plate/Grating Options' and 'Polygon Shape Creation'. In the first section, 'Surface' is set to 'Concrete Slab', 'Thickness' is '6.0000', 'Grade' is 'FC\_5', 'Class' is '1', 'Slab Offset' is '6.000', and 'Naming' is 'Dynamic Naming'. The 'Polygon Shape Creation' section shows 'Status: NONE', '0 Bms/Grids/Lines Selected', '0 Nodes', and '0 Polygon Sides'. There is a button 'Select Beam/Grid/Line 1' and a dropdown 'Outside Edge'. Below this, it says 'Beam Not Selected'. At the bottom, 'Control Elevation' is '0.000 Ft' with a button 'Set Elevation to Last Datapoint'. The bottom of the dialog has four buttons: 'Place Slab', 'Accept', 'Clear all Beams', and 'Cancel'.

Polygon Slab, Plate & Grating on Beams - Primary Dialog Box

## **ACE Polygon Slab, Plate & Grating on Beams Documentation**

The utility has internal defaults for all the items shown on the dialog box above. The defaults may be overridden with user definable defaults by using a definition file, which is discussed in detail later in this document.

# ACE Polygon Slab, Plate & Grating on Beams Documentation

## Polygon Slab, Plate & Grating on Beams - Variables and Options

The Polygon Slab Plate & Grating on Beams application, ACE\_PSG.MA, has been designed to allow greatly facilitate the placement of polygon surfaces.

- Option Button to select surface (Concrete Slab, Steel Plate & Steel Grating) & parameters
  - Option Button to select Grade (per surface type)
    - Up to 10 grade choices
    - Grade choices are user definable via definition file
  - Option Button to select Thickness (per surface type)
    - Up to 9 thickness choices
    - Thickness choices are user definable via definition file
    - A specified thickness may be keyed in at run time
  - Slider/Keyin to select class (0 to 9) (per surface type)
  - Surface Elevation Offset (per surface type)
- Naming Option Parameters
  - Naming can be dynamic, constant, constant w/ FWP ID or FQP autaname
  - Toggle for Mbr\_ID suffix w/ dynamic naming
  - Name Input keyin field for dynamic or constant naming
- Select Beam/Grid/Line (Bm1) through Beam4(Bm30) parameters
  - Button to select a support beam or grid line or line
  - Offset placement option (Outside Edge, Center (CP line), Inside Edge, or Defined Distance) (NOTE only FWP members w/ CP=8 have Outside Edge & Inside Edge options)
  - Defined offset distance if defined distance is selected
  - Button Allowing “Backing Up” after first element selected (i.e. deleting last selection) (NOTE all selected elements may be removed by repeatedly pressing “Clear Beam“ button)
- Control Elevation Definition
  - Keyin field Option or
  - “Last Datapoint” button
- Buttons for “ReDisplay Poly Points” or “ReDisplay Shape”
- Buttons to “Place Slab”, “Accept Polygon”, “Clear all Beams” or “Cancel”

## Surface Naming

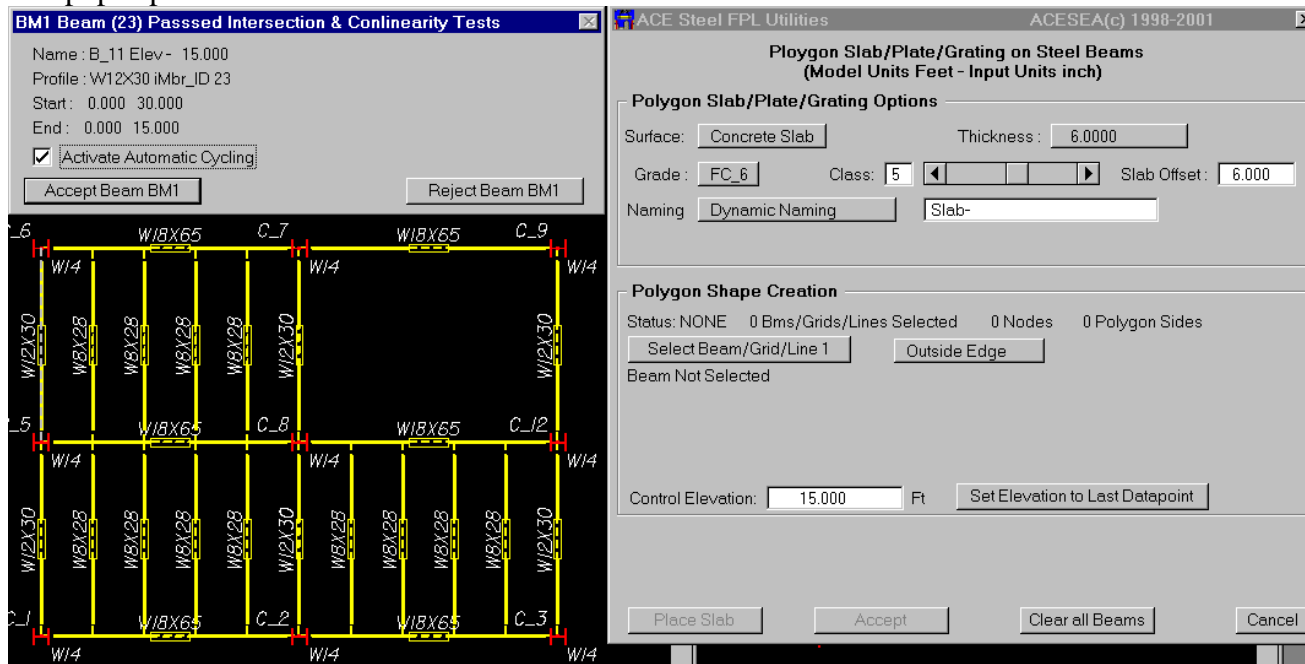
The surface placed consists of one FrameWorks solid (slab type) element and may be named depending upon the naming option selected. The name may be a constant name or it may be a prefix with the FrameWorks solid member ID (FWP ID) appended as a suffix. The default prefix is PSG, however a different prefix may be specified in the definition file or supplied at runtime. Other naming options include: dynamic naming at placement time: a constant specified name; or FrameWorks normal naming for individual components (autaname). Dynamic naming allows the name to be selected (or remain the last name selected) at placement time with or without appending the FWP ID to the name.

Subsequently a sample definitions file is shown with expanded explanation.

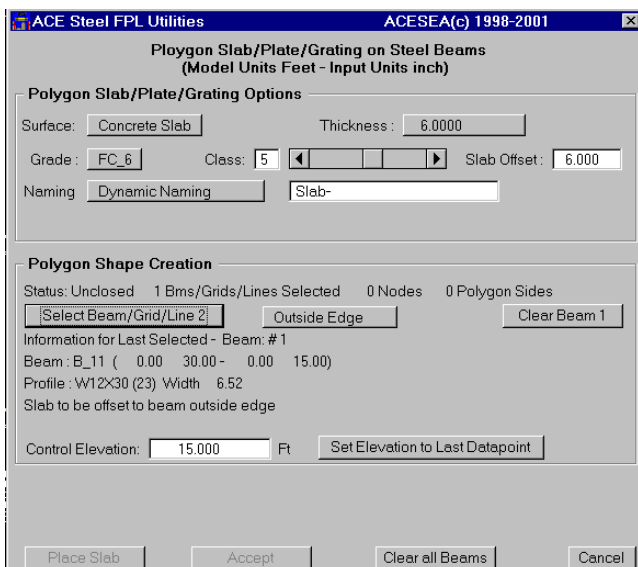
# ACE Polygon Slab, Plate & Grating on Beams Documentation

## Polygon Slab, Plate & Grating on Beams - Basic Operation

The process of placing a surface is initiated by selecting the starting element (beam/grid/line). As previously mentioned the successive elements must project an intersection to the previous element. (For the intersection computation, all elements are projected to the control elevation – i.e. only X & Y coordinates are used in intersection calculation). The element selection progression may be either clockwise or counterclockwise. The detailed element checks are discussed in the next section. The easiest way to start the element selection process is to start by pressing the “Select Beam/Grid/Line 1” button. Select a element and the accept/reject beam dialog box pops up as shown below.



The “Automatic Cycling” is active by default and may be toggled off. With automatic cycling on, the application will expect the selection of elements to continue in progression until the final element is selected with the close polygon toggle option on. For this example an “L” shaped slab will be formed around the 3 bays with interior support beams. After the beam is selected, the primary dialog box looks as follows:

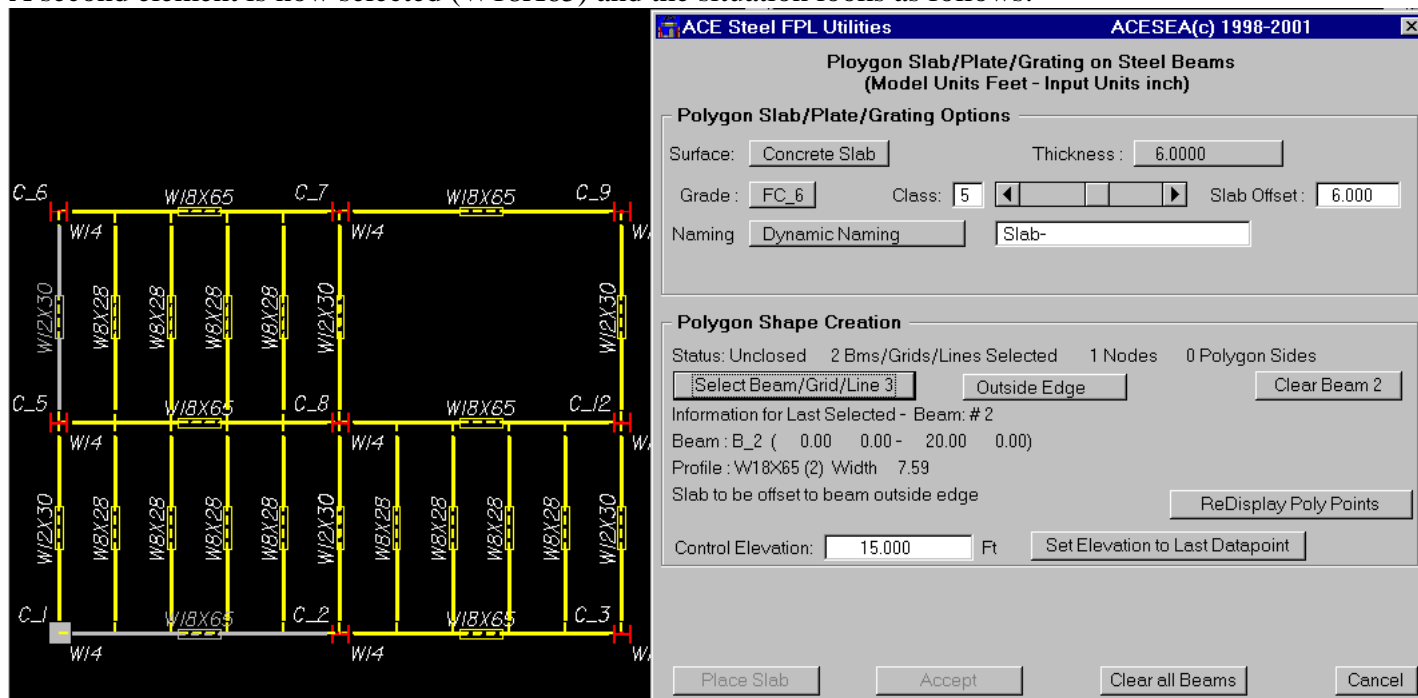


In the figure left, the first element (in this case a W12X30) has been selected and the application is ready for the second element. The first element defines the control elevation but does not define any nodes or sides. However the last side of the polygon will be parallel to the first element selected. Notice that a Button “Clear Beam 1” is now present. If desired the last beam selected can be cleared by pressing this button. In fact all elements selected can be cleared by continuously pressing this button until they are no longer any elements selected. This is effectively removing an element at a time from the back. All elements can be cleared at any time by pressing the “Clear all Beams” button. The edge handling (outside, inside, center or defined) may be selected before the element is selected or may be altered for any element after the polygon is closed.

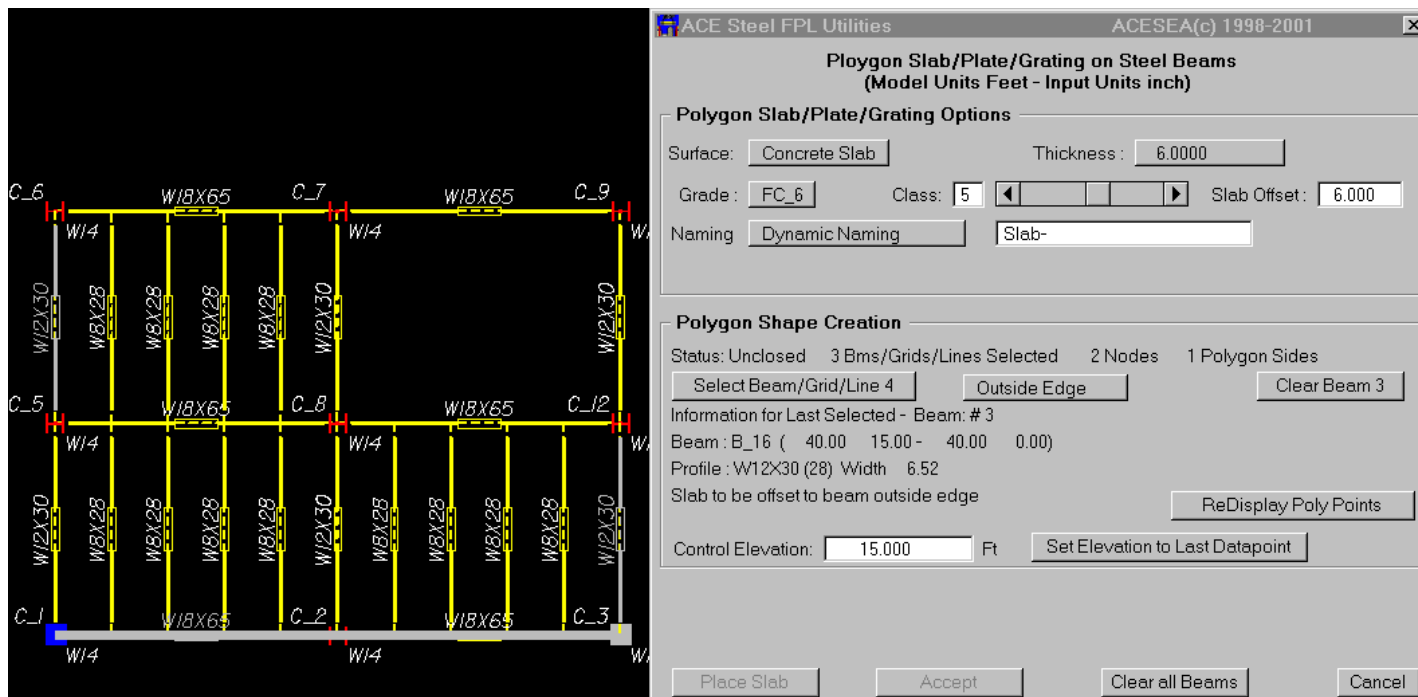
# ACE Polygon Slab, Plate & Grating on Beams Documentation

## Slab, Plate & Grating on Beams/Grids/Lines - Basic Operation (continued)

A second element is now selected (W18X65) and the situation looks as follows:



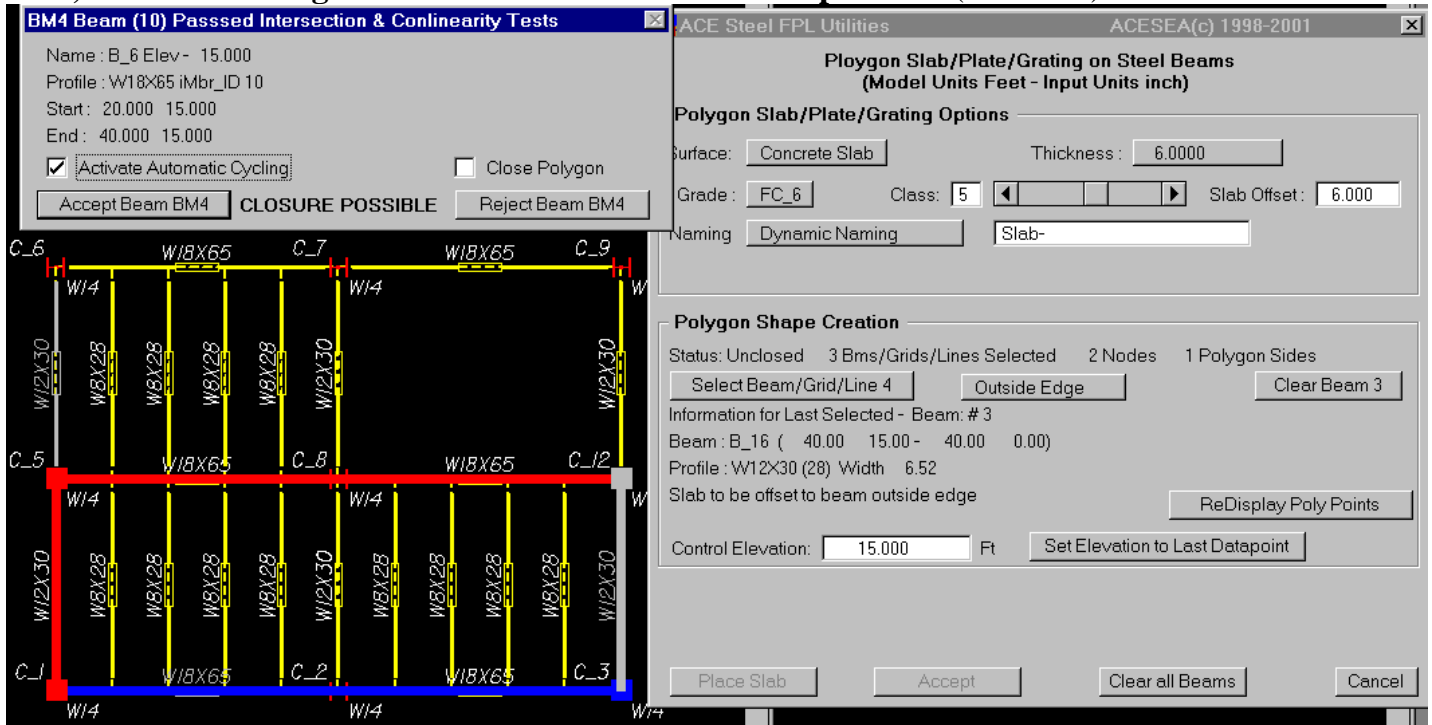
Notice that a node now is shown as a highlighted heavy point. A new button, “ReDisplay PolyPoints” is now present on the primary dialog box. Since the display elements are temporary, the “ReDisplay PolyPoints” is provided to refresh the display. A third element is now selected and the situation looks as follows:



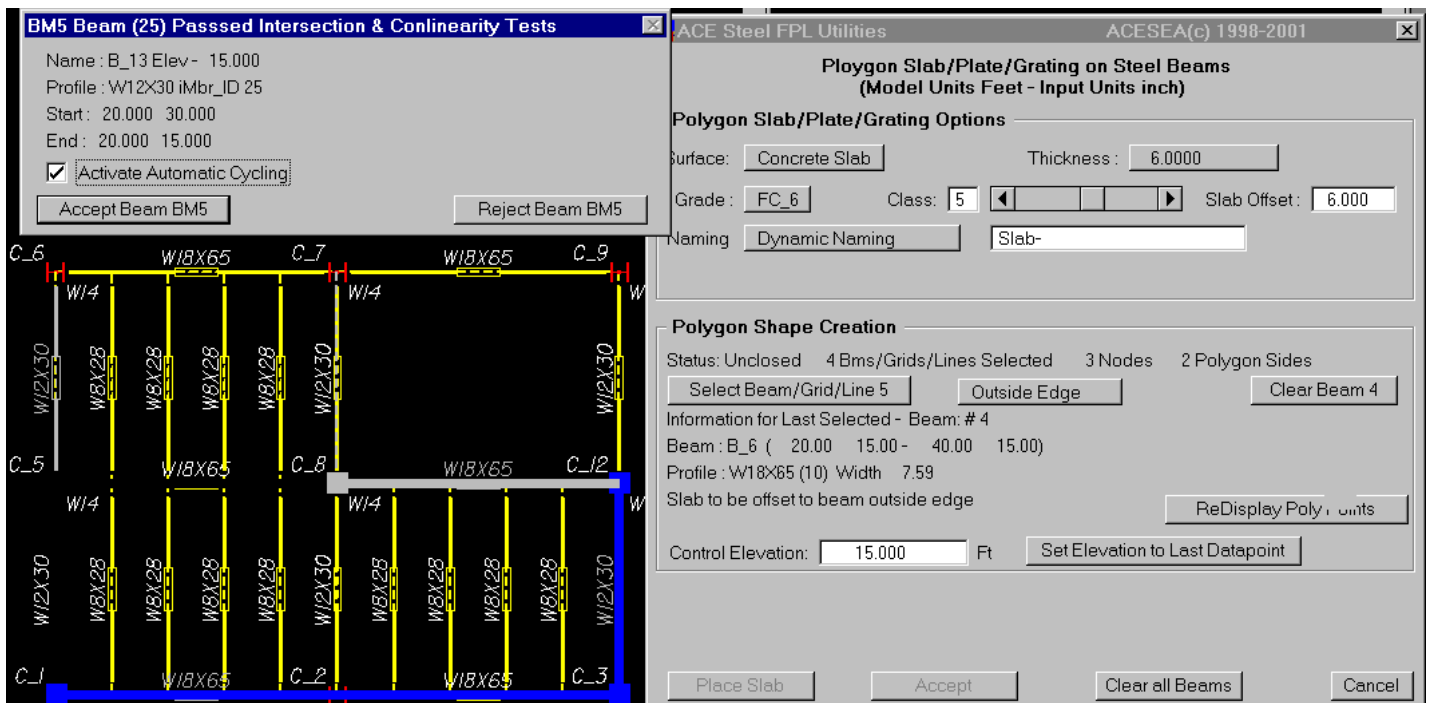
Notice that now there are two points and a polygon side displayed in temporary graphics. The most recent vertex and side is highlighted and previous ones are shown in blue. Next the W18X65 (from C\_6to C\_12) is selected. When this element is selected, an option to close is presented and the display looks as follows:

# ACE Polygon Slab, Plate & Grating on Beams Documentation

## Slab, Plate & Grating on Beams/Grids/Lines - Basic Operation (continued)



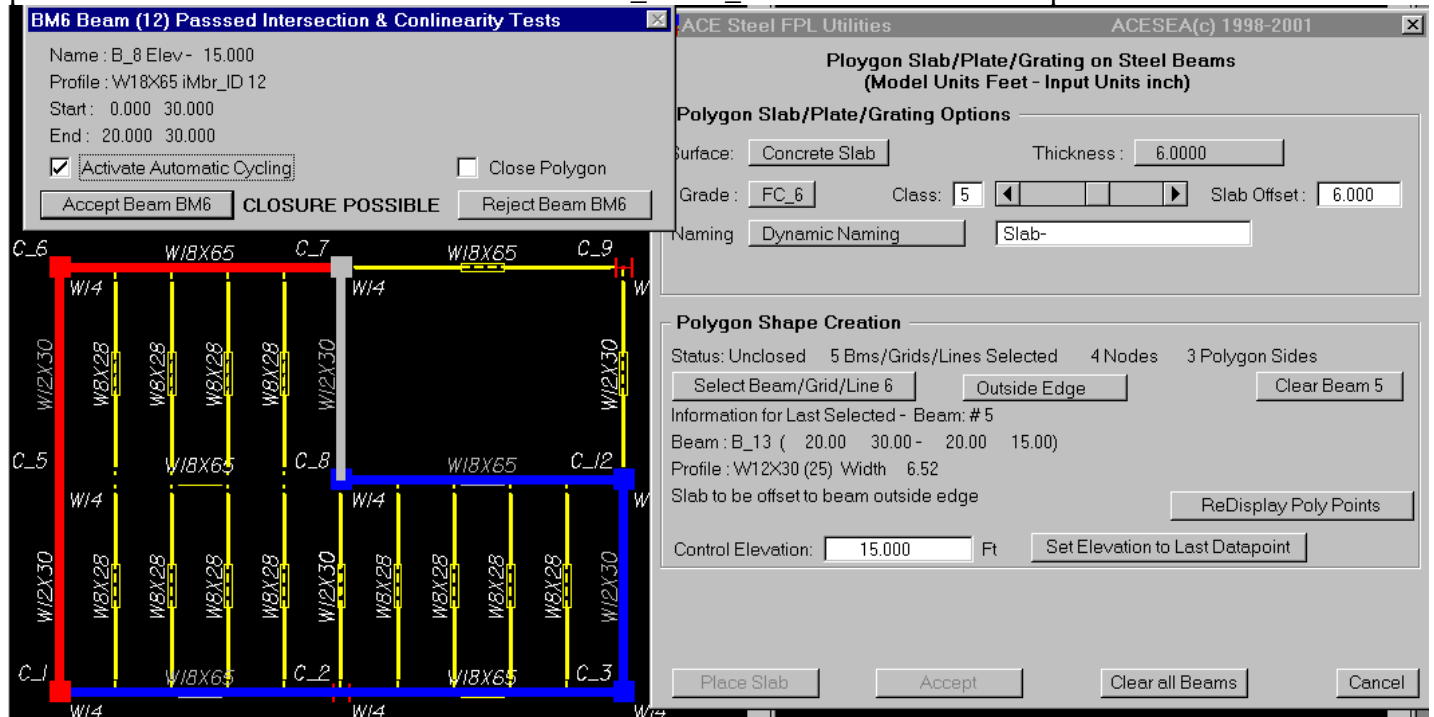
If the “Close Polygon” toggle is checked and the element is selected, the polygon would be closed and the surface would be a four sided polygon show by the blue, highlight and red temporary graphic lines. The red depicts the closure. The blue depicts the now old vertices and sides with acceptance and the highlight depicts the latest vertex and side. Obviously we don’t want to close here as we are forming an “L” shape. So this is accepted with the closure toggle off. Next the W12X30 from C\_6 to C\_7 is selected:



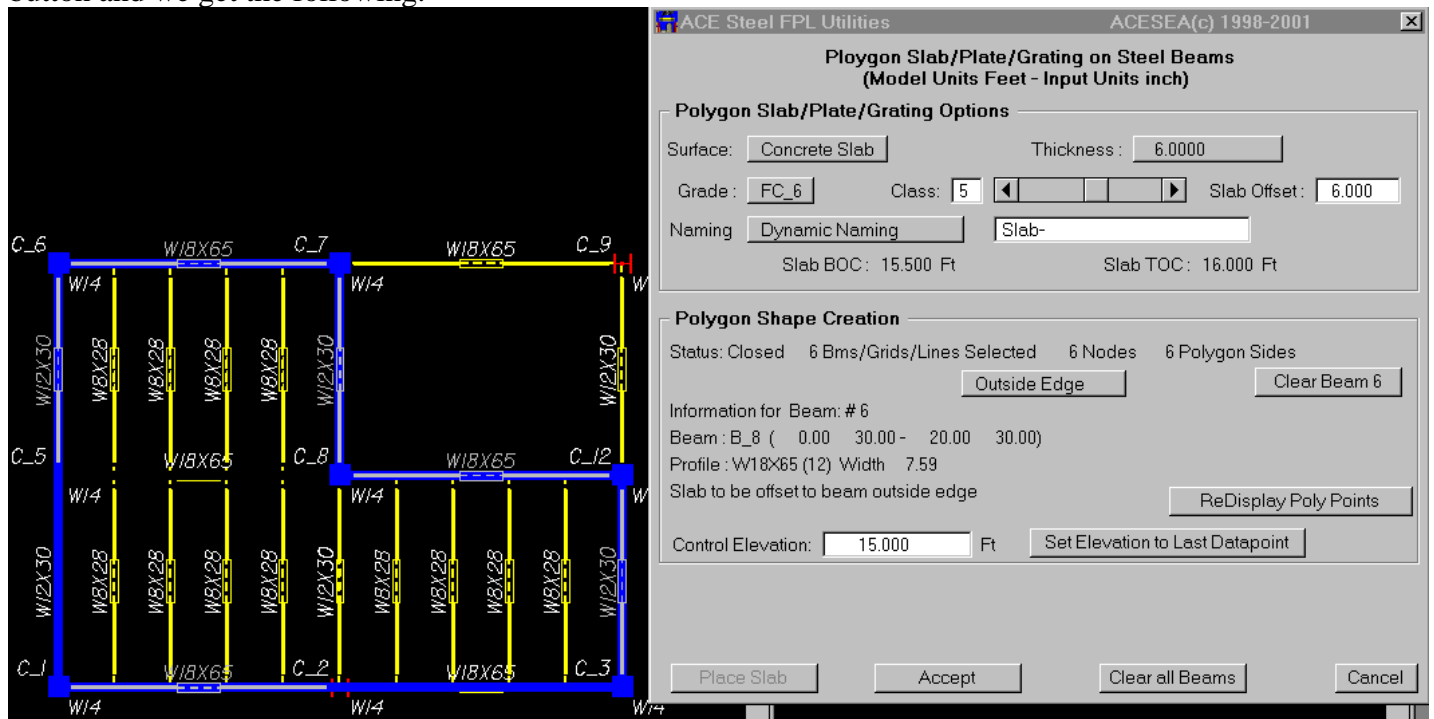
# ACE Polygon Slab, Plate & Grating on Beams Documentation

## Slab, Plate & Grating on Beams/Grids/Lines - Basic Operation (continued)

The last selection did not present a closure option, but now there are 4 nodes and 3 sides determined. The final pick to form the “L” slab is the W18X65 from C\_7 to C\_6. After this element is picked the situation looks like:



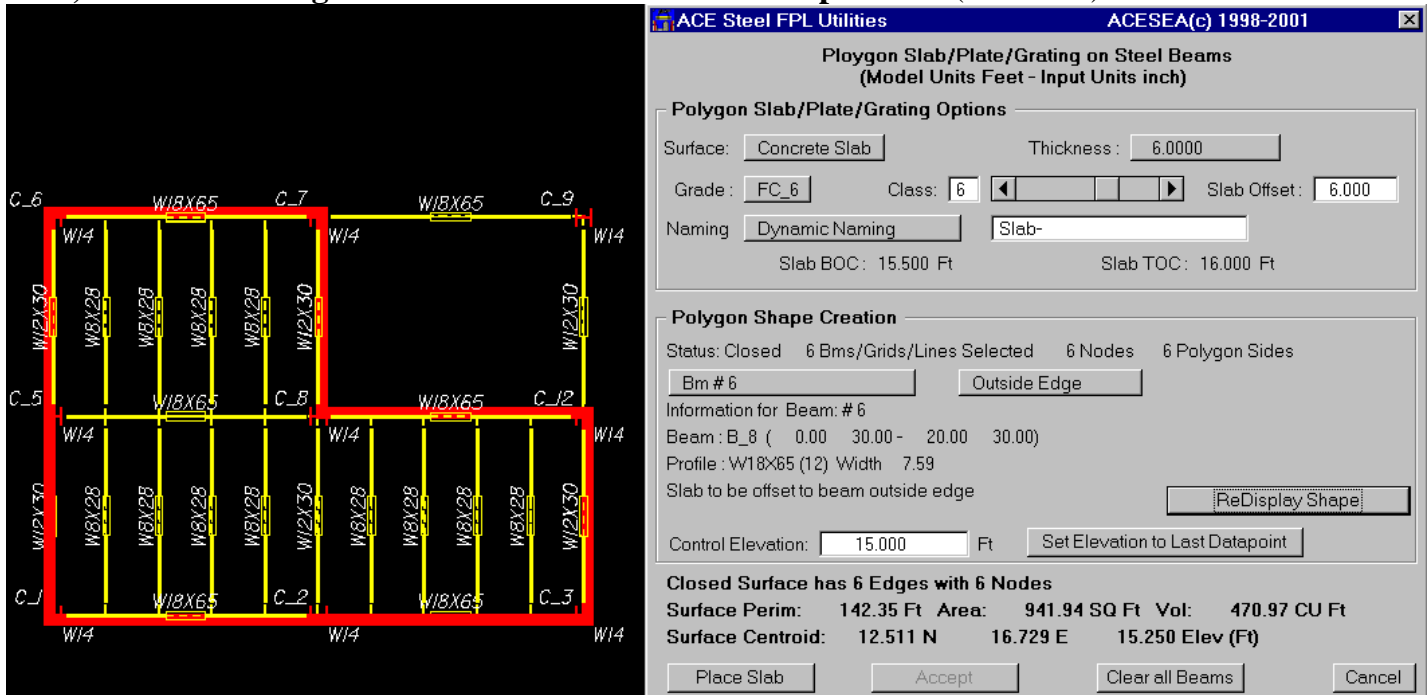
Again a closure option is presented and this time we want to accept it. Turn the toggle on and hit the accept button and we get the following.



At this point, we still have the option to back up by pressing the “Clear Beam 6” button. Since this is the “L” slab we desire, we press the “Accept” button. There is no backing up after accept – the picture looks like:

# ACE Polygon Slab, Plate & Grating on Beams Documentation

## Slab, Plate & Grating on Beams/Grids/Lines - Basic Operation (continued)



Notice that the “L” surface is shown with a red temporary display shape. The element is shown such that it depicts the surface limits (in this case to the outside edge of the beams). Notice that the primary dialog box no longer allows clearing the last beam. All of the beams can be cleared and you would be starting over from scratch. Notice also that the “ReDisplay Poly Points” has changed to “ReDisplay Shape”. Also notice that the “Select Beam/Grid/Line #” button has changed to “Bm #6”. This option button may now be used to change the edge point for the surface for this side. The elevation, class, slab type, thickness, name, grade or offset can be changed as desired. To place this slab, press the “Place Slab” button and the slab is placed. The primary dialog box does not change but remains as is allowing placement of the same slab at another elevation (also the changes mentioned above could be made). To place a totally new surface, press the “Clear All Beams” button.

Virtually any valid polygon with from 3 to 30 sides can be easily placed using the above procedure. The sides need not be orthogonal as with the “L” shape. Shapes such as octagons or stars or whatever can easily be placed. The basic rules for the formation of the polygon are as follows:

### Intersection & Co linearity Checks for beams, grids & lines

Whenever an element (beam/grid/line) is selected, it goes through the following checks.

1. The element must not be vertical (i.e. start & end x & y coordinates may not be equal)
2. Element 2 must intersect element 1 (i.e. element n must intersect element n-1)  
Only X & Y coordinates are used to compute intersections as all members are projected to a flat plane at the Control Elevation. The resulting surface will always be a flat surface (Equal Z).
3. The last element must intersect first element.
4. Element 2 must not be collinear with element 1 (i.e. element n must not be collinear with element n-1)
5. The last element must not be collinear with the first element.
6. Formed polygon sides may not cross each other.



# ACE Polygon Slab, Plate & Grating on Beams Documentation

## Definitions File

Due to the dissimilar nature of the variables in the steel utilities, each steel utility has a separate definition (DEF) file. While each file is distinctly different, each file is similar in the basic method of definition. Each definition file may optionally be controlled with either of two environment variables. Thus a project specific definition file for each project may be easily specified. The environment variables may be specified in numerous ways (similar to any MicroStation variable), however the utilization of a project.pcf is highly recommended. Environment variable definition is discussed in detail in the installation notes provided with the ACE FrameWorks utilities. The default name and location for the definition file for this utility are: C:\ACE\_PSG.DEF. A directory for the definition file may be specified with the environment variable ACE\_DEF\_PATH (will look in specified path for file ACE\_PSG.DEF). A complete name and location of a definition file may be also specified with the environment variable ACE\_PSG\_DEF. The first valid definition file found is utilized. The search for a definition file happens in the following order or priority:

1. If the variable ACE\_PSG\_DEF is specified, the named file at this location will be used if found.
2. If the variable ACE\_DEF\_PATH is specified and ACE\_PSG.DEF is found in this directory, it is used.
3. If there is a c:\ace\_psg.def file it is utilized.
4. If none of the above, internal program defaults are utilized – a warning message will be displayed. (if environment variables in 1 and/or 2 above are specified and corresponding DEF file is not found, a warning is displayed)

A sample default file is provided in later sections of this document. Toward the end of this document, the commands for the definition file are outlined in detail.

Due to the complex nature of the variables, the steel definition files allow the specification of units (either Metric (mm) or English (inch) for this application). Thus a given default file may be utilized in either a Metric or English project. The units may be changed throughout the definition file. If units are not specified, it is assumed that the definition file units match the units of the model (feet/inch-English & meters/mm-Metric). If units are defined and they do not match the model, the variables after the units command are converted to match the model units.

## Sample Definitions File

### Typical Definition File :

```
UNIts  ENGLISH
GRA SLA 4      FC_3      FC_4      FC_5      FC_6
GRA PLA 3      A36       A42       A50
GRA GRA 1      A36
TKS SLA 5      3         4         5         6         8
TKS PLA 3      .25       .375      .5
TKS GRA 1      1.0
BMS 1         6.0
SUR SLA FC_3    1      5.0  6.0
SUR PLA A42     2      .375  0.0
SUR GRA A36     3      1.5  .125
NAME      DYNamic      Slab-
```

# ACE Polygon Slab, Plate & Grating on Beams Documentation

## Definitions File - Command Definition

- Valid Primary Keyword Commands: (UNI, BMS, GRA, TKS, SUR, NAME)
- Each record must begin with a valid primary keyword or it is ignored
- All records that start with a blank are considered comments
- The commands/keywords (records) may be placed in any order however the order is significant
- All values for a given command must be defined in order shown above. If default values are acceptable, only the changed values must be given. However all values up to that point must be defined whether changed or not.
- The components of a given command (record) must all be present and in the order shown
- The units command is special and may be repeated and located as required. While commands may be in any order, it should be obvious that the location of the units command is extremely important.
- By default application looks for C:\ACE\_PSG.DEF definition file
- Definition file path may be defined with environment variable ACE\_DEF\_PATH
- ACE\_DEF\_PATH=d:\mydir\  
(the DEF file ACE\_PSG.DEF will be looked for in the directory d:\mydir)
- Definition file may be defined with environment variable ACE\_PSG\_DEF
- ACE\_PSG\_DEF = d:\mydir\mydef\_file (*highest priority definition*)  
(the DEF file mydef\_file will be looked for in the directory d:\mydir)
- NOTE: Components shown in bold may only be specified in the definitions file

## UNIT Command - Units Command (optional command)

**UNIT** {UNITTYPE}

where :

{UNITTYPE} May be ENGLISH or METRIC

All input for the application is in inches for English units and mm for metric units.

If units is not specified it is assumed that the units match the current model units.

Units may be changed at any time but be aware that the properties (in attached library) for the member specified must match the current model units.

## BMS Command - Beam Support Command defines support beam parameters

**BMS** *iPlace\_opt* *fDistance*

where :

**iPlace\_opt** : Default placement option ( valid options 1 - 4)  
1 - Outside Edge  
2 - Centerline (CP line)  
3 - Inside Edge  
4 - Defined

*NOTE: Outside Edge & Inside Edge only available with FWP members w/ CP = 8  
Other members, FWP grids & MicroStation lines will default to Centerline (CP line)*

**fDistance** : Default distance to surface edge to beam/grid centerline when defined  
(a positive value will always increase both the surface area and volume of the surface)

# ACE Polygon Slab, Plate & Grating on Beams Documentation

## Definitions File - Command Definition (con'd)

### GRA Command - Grade Command defines grade options

**GRA** *sSurface iGrades sGrade1 ... sGraden*

where

- sSurface** : Surface type : **SLA** (Concrete Slab), **PLA** (steel plate) or **GRA**(steel grating) (default value - SLA)
- iGrades** : The number of grade values for this surface type (Maximum of 10 grades).  
(default value - 3,2,2 for surfaces SLA, PLA, GRA)
- sGrade1** : Grade definition (24 character max - should also be defined in FrameWorks)
- sGraden** : Last grade definition (Maximum of 10 grades n <= 10).

### TKS Command - Thickness Command defines thickness options

**TKS** *sSurface iTks sTks1 ... sTksn*

where

- sSurface** : Surface type : SLA (Concrete Slab), PLA (steel plate) or GRA(steel grating) (default value - SLA)
- iTks** : The number of thickness values for this surface type (Maximum of 9 thickness values).  
(default value - 3,2,2 for surfaces SLA, PLA, GRA)
- fTks1** : First thickness definition
- fTksn** : Last thickness definition (Maximum of 9 thickness values n <= 9).

### SUR Command - Surface Command defines standard values for surfaces

**SUR** *sSurface sGrade iClass fTk fOffset*

where

- sSurface** : Surface type : SLA (Concrete Slab), PLA (steel plate) or GRA(steel grating) (default value - SLA)
- sGrade** : Default Grade value (must match value in GRA definition or 1<sup>st</sup> Grade option is displayed)
- iClass** : Default Class value ( 0 <= iClass <= 9)
- fTk** : Default Thickness (fTK > 0) (if matches a thickness definition it is displayed)
- fOffset** : Distance from Beam/Grid TOS to surface BOC/BOS (Positive is up)

### NGP Command - Named Group Command defines namedgroups

(Optional command to define named groups)

**NGP** *iNGP\_slab iNGP\_plate iNGP\_grating*

where

- iNGP\_slab** : Named group for slab surfaces ( default -1 which is none)
- iNGP\_plate** : Named group for plate surfaces ( default -1 which is none)
- iNGP\_grating** : Named group for grating surfaces ( default -1 which is none)

NOTE: Namedgroups are defined globally for a project. The iNGP\_xxx value is an integer value that corresponds to the index of the global namedgroups. The first namedgroup is 0, the next is 1 and so on up to a maximum integer value of the number of namedgroups minus one. If a name group does not exist for the integer value specified, the member type in question will simply not be placed in a named group. A value of -1 specifies that the member type in question is not to be put in a namedgroup. In FWP namedgroups are specified by an alpha name so be careful when selecting integers. **SOLID NAMEDGROUPS ARE FUNCTIONAL with FWP version 7.00.00.17 and later.**

# ACE Polygon Slab, Plate & Grating on Beams Documentation

## Definitions File - Command Definition (con'd)

### NAME Command - Name Command defines method of naming components

**NAME** {*NAME\_OPTION*} *name\_prefix*

where

{NAME_OPTION}	: Keyword - must be AUT or SPE or DYN or CON
<b>SPE</b> cified	: Use the supplied name and append the member ID for surface placed Thus each surface will have a different name (This is the default option with the name "SPG")
<b>DYN</b> amic	: At placement time will display last name used with following options 1) option to supply a new name 2) option to request that member ID for surface placed be appended Thus each surface will have a different name 3) option to abort placement
<b>AUT</b> o	: FrameWorks assigns names by type and sequence number (name_prefix not required or utilized)
<b>CON</b> stant	: Use this name for all surfaces

# **ACE Polygon Slab, Plate & Grating on Beams Documentation**

## **LOG FILES**

All applications can write log files if the environment variable ACE\_DUMP is set to 1. There have been reports that some sites lock the C root drive and under certain conditions a locked C drive can cause a system fault 5.

All applications have been modified to warn of a locked drive/file and then gracefully exit. All applications now look for the environment variable ACE\_LOG\_PATH. If it is found, that is the directory where the log files will be placed. If the directory is locked or non-existent or if file is locked a warning will be given and the C drive will be tried. If it is locked or the file is locked a warning will be given and application will gracefully exit.

Usage of the variable ACE\_LOG\_PATH to control log file locations is similar to ACE\_DEF\_PATH to control DEF files. However there is one very important difference: ACE\_LOG\_PATH should NEVER point to a network drive (this is highly recommended for ACE\_DEF\_PATH). Everyone writes to the same named log file and if they are on a network drive there will be bad consequences. ALWAYS point ACE\_LOG\_PATH to a local drive (perhaps a temp off C root).