Appendix-C

Commands

AM2DPATH

This command is used to solve (profile) the 2D entities and create 2D path for sweeping the profiles.

AM2SF

This command is used to convert the selected designer model, AutoCAD solid, 3D meshes, lines, arcs, circles and polylines with thickness into 3D AutoSurf surfaces.

AM3DPATH

This command is used to create 3D paths for sweeping the profiles. You can create helical path, spline path or pipe path using this command. You can also create a 3D path from an existing edge using this command.

AMABOUT

This command is used to display the information regarding the version and copyright of the Mechanical Desktop.

AMACISOUT

This command is used export the designer models or assemblies into an ACIS (.SAT) file. The version of the ACIS file is controlled using the AMACISOUTVER system variable.

AMACTIVATE

This command is used to activate a local or external designer parts, subassemblies, assemblies or the scenes.

AMADDCON

This command is used to apply the parametric constraints to a closed sketch, an open entity, cut line, split line or a break line.

AMADJUSTSF

This command is used to close the gap between the two selected surfaces by adjusting the edges of one or both the surfaces.

AMANALYZE

This command is used to analyze the selected surface. You can perform the analysis for curvature, draft or reflection lines using this command.

AMANGLE

This command is used to apply the angle assembly constraint to the selected local or external designer part or a subassembly.

AMANNOTE

This command is used to create or modify parametric annotations in the drawing views.

AMASSIGN

This command is used to create or modify the user-defined annotations to the selected designer part.

AMASSMPROP

This command is used to calculate the mass properties of the components of selected assembly. You can write the result of this command into a .MPR file.

AMATTACHSYM

This command is used to attach or display the symbols.

AMAUDIT

This command is used to find out whether or not all the external parts in the current drawing are updated.

AMAUGMENT

This command is used to create the augmented lines on the selected surface edge or the display line.

AMAUTODIM

This command is used to automatically dimension the drawing views in the layouts.

AMBALLOON

This command is used to add the parametric callouts to the part references in the drawing views in layouts.

AMBASICPLANES

This command is used to place the work planes at the top, front and side views of a specified point. A new part definition is created when you place the work planes using this command.

AMBEND

This command is used to bend an existing feature using an open profile.

AMBLDFEATNM

This command is used to list the features displayed in the desktop browser in the language of the last loaded version of Mechanical Desktop.

AMBLEND

This command is used to create a smooth surface between two, three or four wires.

AMBOM

This command is used to create or edit the BOM database to be used in the part lists in drawing views.

AMBREAK

This command is used to break the selected surface into two about the specified U or V edge.

AMBREAKLINE

This command is used to create the break lines to be used for generating the breakout section views in the layouts.

AMBROWSER

This command is used to control the display of the desktop browser.

AMBROWSERFONT

This command is used to control the type and size of the fonts used to list the features in the desktop browser.

AMCATALOG

This command is used to recall, attach or detach the external parts or subassemblies. You can localize the part or keep is as an external part in the drawing.

AMCENLINE

This command is used to create the parametric centerlines or center marks in the selected circles or arcs in the drawing views.

AMCHAMFER

This command is used to bevel the edges of

the designer model. The edges can be bevelled using one distance, two distances or one distance and one angle.

AMCHECKFIT

This command is used to check the minimum 3D distance or maximum 3D deviation between a surface and a wire or between two wires. This is done to check the product design and data for Coordinate Measuring Machines.

AMCOMBINE

This command is used to create complex designer models by applying the parametric boolean operations on them. The valid boolean operations that can be performed using this command are union, subtraction and intersection.

AMCOPYFEAT

This command is used copy the features within a designer model or from one designer model to the other. The new feature will be placed on the current sketch plane.

AMCOPYIN

This command is used to copy the external parts in the current drawing. However, this command localizes the parts placed.

AMCOPYOUT

This command is used to copy out the selected designer part into a new drawing file.

AMCOPYSCENE

This command is used to create a new scene by copying an existing scene.

AMCOPYSKETCH

This command is used to copy an unconsumed sketch or the sketch of a feature.

AMCOPYVIEW

This command is used to copy the selected drawing view to a new location in same layout or to a different layout.

AMCORNER

This command is used to create a blended surface at the corner of three filleting surfaces.

AMCUTLINE

This command is used to create the parametric offset or aligned cut lines for generating the offset or aligned section views in the layouts.

AMDATUMID

This command is used to create the datum identifiers in the form of frames. The drafting standards that can be used are ANSI, BSI, DIN, ISO, JIS, CSN, or GB.

AMDATUMTGT

This command is used to place the datum targets in the drawings. The drafting standards that can be used are ANSI, BSI, DIN, ISO, JIS, CSN, or GB.

AMDELCON

This command is used to delete the constraints applied to the profiled sketches.

AMDELETE

This command is used to delete the designer parts, instances or scenes.

AMDELFEAT

This command is used to delete the selected feature. If the selected feature has some dependent features, they will also be deleted.

AMDELTRAIL

This command is used to delete the trails from the exploded assemblies in the **Scene** mode.

AMDELTWEAKS

This command is used to delete the tweaks from the exploded assemblies in the **Scene** mode. The component moves back to its original position once the tweak is deleted.

AMDELVIEW

This command is used to delete the specified

drawing view from the layout. If the selected view has some dependent views then you will be asked whether you want to delete the dependent views or retain them.

AMDIMALIGN

This command is used to align the selected dimension in the drawing view with a specified base dimension.

AMDIMARRANGE

This command is used to arrange the dimensions in the drawing views or in the sketches in the **Model** tab.

AMDIMBREAK

This command is used to break the dimension line or the extension line in the drawing views. The associativity of the dimensions is not lost by breaking them using this command.

AMDIMDSP

This command is used to control the display mode of the dimensions. The dimensions can be displayed as parameters, equations or as numbers using this command.

AMDIMFORMAT

This command is used to modify the dimension format for the linear, angular, radial or diameter dimensions in the drawing views.

AMDIMINSERT

This command is used to break an existing dimension into two individual dimensions. The break point for the dimensions is specified by you.

AMDIMJOIN

This command is used to join two individual dimensions into a single dimension. This reduces the number of dimensions in the drawing views.

AMDIMMEDIT

This command is used to edit multiple dimen-

sions in a single attempt.

AMDIRECTION

This command is used to display and reverse the direction of the selected surface. This is used in the NC machining or in the Coordinates Measuring Machines.

AMDISPSF

This command is used to control the display of the AutoSurf surfaces. You can change the surface normal size, the number of the U and V display lines and the surface patches using this command.

AMDIST

This command is used to calculate the minimum 3D distance between the selected components in the 3D space during the analysis of the assembly.

AMDWGDIMDSP

This command is used to control the display of the parametric dimensions in the drawing views. You can display the parametric dimensions in the drawing views as parameters, equations or numbers.

AMDWGVIEW

This command is used to create the 2D drawing views from the designer models or the assembly scenes. You can create orthographic, auxiliary, isometric, section, detail and broken views using this command.

AMEDGE

This command is used to untrim, copy and extract the edges of the surfaces. This command is also used to display the edge nodes at the surfaces and the faces.

AMEDGESYM

This command is used to generate a symbol that is used to define the current condition of the selected edge during the annotation of the drawing using the DIN standard.

AMEDIT

This command is used to edit the annotations like welding symbols, feature control systems, surface textures and datums.

AMEDITAUG

This command is used to resize, rotate, copy and modify the vectors on the augmented lines. You can also create augmented lines from lines and polylines using this command.

AMEDITCONST

This command is used to modify the distance and angle offset value of the assembly constraints. You can also delete the individual or all the assembly constraints applied to the assembly.

AMEDITFEAT

This command is used to edit the existing features on the designer models.

AMEDITSF

This command is used to modify the surfaces. Various modifications that can be performed using this command are lopping off the surfaces, reverse the direction of the surface normal, increasing the number of the surface grips, changing the density of the surface grips in the U and V directions and changing the distance between the surface grips.

AMEDITTRAIL

This command is used to edit the trails added to the exploded scenes of the assemblies in the **Scene** mode.

AMEDITVIEW

This command is used to modify the scale factor, associativity and the hidden lines display options of the drawing views created in the layouts.

AMEXTRUDE

This command is used to create 3D designer model from a profiled sketch by extruding it in the Z direction of the current sketch plane.

AMEXTRUDESF

This command is used to create the surface by extruding the lines, polylines, arcs, circles, ellipses or splines.

AMFACEDRAFT

This command is used to apply the draft angle to the faces of the designer model for its easy withdrawal from the molds during manufacturing.

AMFACESPLIT

This command is used to split the selected faces of the designer model using a planar face or by projecting a split line on to the selected faces.

AMFCFRAME

This command is used to create the feature control frames used to display the tolerances, orientation, position and so on. The standards that can be used are ANSI, BSI, CSN, DIN, GB, ISO, and JIS.

AMFEATID

This command is used to create the feature identification symbol to be used in the feature control frames. The standards that can be used are ANSI, BSI, CSN, DIN, GB, ISO, and JIS.

AMFILLET

This command is used to fillet the edges of the designer model. The fillet feature created on the designer model in parametric in nature and can be edited at any point of time.

AMFILLET3D

This command is used to fillet the selected wires in the current drawing. The wires that can be used for this command are lines, coplanar polylines, arcs, elliptical arcs, circles, ellipses and splines.

AMFILLETSF

This command is used to fillet any two selected

surfaces. The fillet surface thus created will be an independent surface.

AMFITSLIST

This command is used to copy the existing fits and their dimension values in a list that can be copied in the drawing. This command is also used to update the fits list when the fits are modified.

AMFITSPLINE

This command is used to fit the spline into a selected polyline, arc, circle or an ellipse and at the same time retaining the properties of the original entity. The spline can be fitted either using the tolerance value or the control points.

AMFLOW

This command is used to create the 3D polylines or the augmented lines to be displayed on the surfaces in the U and/or V directions or on the C1 tangencies.

AMFLUSH

This command is used to apply the flush assembly constraint to the components for placing them parallel to each other.

AMHOLE

This command is used to place a parametric drilled, counterbore or countersink hole on the selected planar face of the designer model.

AMHOLENOTE

This command is used in the **Drawing** mode to automatically extract and display the information related to the selected hole in the drawing view.

AMIDFIN

This command is used to convert a printed circuit board data in the ID format (Intermediate Data) into a Mechanical Desktop entity.

AMINSERT

This command is used to apply the insert

assembly constraint to the two selected components with circular edges or faces. This constraint allows both the selected circular faces or edges to share same central axis and at the same time allows them to make their faces coplanar. You can also specify some offset value for the selected faces.

AMINTERFERE

This command is used to analyze the components of the selected assembly for interference. You have an option of creating the interference solid that can be used for the analysis of the extra material. This command can also be used for the combined designer parts.

AMINTERSF

This command is used to intersect the selected surfaces at their intersection. You can also create a polyline at the intersection of the selected surfaces.

AMJOIN3D

This command is used to join the selected lines, polylines, ellipses, arcs, circles, splines or augmented lines into a single polyline, spline or augmented line. The new entity will have the properties of the first object selected to join.

AMJOINSF

This command is used to join the selected surfaces. You can join two or more surface using this command.

AMLAYER

This command is used for setting the layer properties in the drawing. When you invoke this command, the **Layer Control** dialog box will be displayed. You can set the layer properties and control the layer groups using this dialog box.

AMLENGTHEN

This command is used to increase or decrease

the length of the selected surface by a specified distance or percentage.

AMLIGHT

This command is used to control the ambient and the direct light that falls on the designer model. When you invoke this command, the lights window appears next to the desktop browser. Using this window you can modify the ambient and the direct light.

AMLIGHTDIR

This command is used to change the target of the current light source that falls on the designer model.

AMLISTASSM

This command is used in the **Model** mode to display the information related to the selected designer parts or subassemblies. The information is displayed in the AutoCAD Text Window.

AMLISTPART

This command is used to list the selected part or all the parts in the **Model** mode.

AMLISTVIEW

This command is used to display the information related to the selected view in the **Drawing** mode. The information is displayed in the AutoCAD Text Window.

AMLOCKSCENE

This command is used in the **Scene** mode to lock or unlock the selected scene. You can not modify the explosion factor and tweaks of the locked scene.

AMLOFT

This command is used to create the complex designer models by blending together two or more dissimilar sketches of faces of the designer models.

AMLOFTU

This command is used to create the surfaces

using a single set of base wires.

AMLOFTUV

This command is used to create a surface using two sets of base wires. The two sets of wires represent the surface in the U direction and in the V direction.

AMMAKEBASE

This command is used to convert the selected designer model comprising of any number of features into a single base feature. No data related to the previous features remain in the memory of Mechanical Desktop and therefore, you cannot edit the base feature.

AMMATE

This command is used to apply the mate assembly constraints to the selected components of the assembly.

AMMIGRATEB

This command is used to convert the data of the previous release of Genius software so that they can be used in the Mechanical Desktop 4.

AMMIRROR

This command is used to create a new designer part by mirroring an existing designer part. You have to specify the name of the new part.

AMMODDIM

This command is used to modify the parametric dimensions of the sketches in the **Model** mode or on the drawing views in the **Drawing** mode.

AMMODE

This command is used to toggle between the **Model** or the **Drawing** mode.

AMMOVEDIM

This command is used to move the selected dimension within the view or from one view to the other.

AMMOVEVIEW

This command is used to move the selected

view to a new location within the current layout or to another layout. If the selected view has some dependent views then they will also move along with the parent view.

AMMANIPULATOR

This command is used to dynamically rotate or move the selected object in the X, Y or Z axis or in the 3D space. You can also create the instances of the selected object at a specified angle or distance using this command.

AMNEW

This command is used to create a new designer part, subassembly or a scene. This command is also used to convert an AutoCAD solid into a designer model.

AMNOTE

This command is used to create an associative note for the parts, holes or the undercuts in the drawing views.

AMOFFSET3D

This command is used to offset the selected 3D polyline.

AMOFFSETSF

This command is used to create a new surface by offsetting the selected surface.

AMOPTIONS

This command is used to set preferences for the designer parts, assemblies, surfaces, scenes, drawing views and annotations in the current drawing.

AMPARDIM

This command is used to add the parametric dimensions to the profiled sketch. All the dimensions added using this command are displayed during the editing of the designer model.

AMPARTEDGE

This command is used to create a line, arc or

a combination of both on the selected edge or face of an existing designer model. The new line or arc can be used for creating a new sketch.

AMPARTLINE

This command is used to create a 3D polyline representing the parting on the selected surface.

AMPARTLIST

This command is used to create a part list in the **Drawing** mode that displays the information related to all the part of the assembly. The information is used from the BOM database.

AMPARTPROP

This command is used to calculate the mass properties of the active designer model. You can also write the result of this command into a .MPR file.

AMPARTREF

This command is used to add the attributes to the nonparametric parts or a specified point. The attributes thus added can also be displayed in the Bill Of Material.

AMPARTREFEDIT

This command is used to edit the attributes added to the nonparametric parts or points using the **AMPARTREF** command.

AMPARTSPLIT

This command is used to split the active designer part into two using a planar face, work plane or a split line.

AMPATTERN

This command is used to create the rectangular, polar or axial patterns.

AMPATTERNDEF

This command is used in the **Model** mode to assign different hatch pattern to all the components of the assembly to be used in the section views in **Drawing** mode. This is done to easily identify various components in the section views.

AMPLANE

This command is used to create planar surfaces. The surfaces can be created by defining two corners of a window enclosing the surface or select a closed wire. If you select more than one wire, the trimmed surfaces are created depending upon the boundaries of the selected objects.

AMPOWERDIM

This command is used to apply the parametric dimensions or nonparametric dimensions to the selected entities in the **Model** mode or in the **Drawing** mode. When applied to an unprofiled sketch in the **Model** mode or to the entities in the drawing views in the **Drawing** mode, these dimensions will be nonparametric in nature and when applied to the profiled sketch in the **Model** mode, these dimensions will be parametric in nature.

AMPOWEREDIT

This command is used to edit the parametric, nonparametric or reference dimensions in the **Model** or the **Drawing** mode. You can also edit the entities like Part Lists, Balloons, Datum Identifiers, Welding Symbols and so on using this command.

AMPOWERSNAP

This command is used to create four personalized snap settings in addition to the current snap settings. When you invoke this command, the **Power Snap Settings** dialog box will be displayed.

AMPRIMSF

This command is used to create the primitive surfaces like cone, sphere, cylinder or torus.

AMPROFILE

This command is used to solve a closed entity

and apply the geometric constraints to it. You can also profile an open entity using this command provided the gap or the opening is very small.

AMPROJECT

This command is used to project the selected wire on the specified surface and output can be obtained in the form of a trimmed surface, polyline at the intersection area or an augmented line at the intersection area.

AMPROJECT2PLN

This command is used to project a selected 2D planar entity or a 3D planar face on the specified work plane, sketch plane or a 3D planar face.

AMPSNAP1

This command is used to load the power snap 1 settings set using the **AMPOWERSNAP** command.

AMPSNAP2

This command is used to load the power snap 2 settings set using the **AMPOWERSNAP** command.

AMPSNAP3

This command is used to load the power snap 3 settings set using the **AMPOWERSNAP** command.

AMPSNAP4

This command is used to load the power snap 4 settings set using the **AMPOWERSNAP** command.

AMRECOVER

This command is used to check the Mechanical Desktop file for the errors and correct the errors, if any, in the selected file.

AMREFDIM

This command is used to add reference dimensions to the entities in the drawing views.

AMREFINE3D

This command is used to refine the selected line or polyline that is used for fitting the surfaces by changing the density of the points on the lines or polylines.

AMREFINESF

This command is used to refine the selected surface by changing the number of U and V patches.

AMREFRESH

This command is used to refresh all the external components in the current drawing if they have been modified since their last reference in the current drawing or the execution of the last **AMREFRESH** command.

AMRENAME

This command is used to rename the selected designer part, instance, scene or the drawing view.

AMREORDFEAT

This command is used to change the order of the feature used in the creation of the designer model.

AMREPLACE

This command is used to replace the external component referenced in the current drawing with other referenced components.

AMREPLACEDGE

This command is used to replace the edge of a selected surface with that of a specified surface. This command not only replaces the edge but also trims the surface at the apparent intersection and blends the transition between the two surfaces.

AMREPLAY

This command is used to display all the steps used in the creation of the selected designer part in the sequence in which they were used. You can truncate the steps at any point of time or suppress the further steps using this command.

AMRESTRUCTURE

This command is used to move the selected designer part or subassembly within the assemblies.

AMREVOLVE

This command is used to create a designer model or a new revolved feature by revolving a profiled sketch about a line included in the sketch, a work axis or an edge of the existing designer model.

AMREVOLVESF

This command is used to create a revolved surface by revolving the base curve about a selected axis.

AMRIB

This command is used to convert an open profile into a rib feature.

AMROTCENTER

This command is used to find out a new center of rotation for all the visible objects.

AMRSOLVESK

This command is used to resolve the selected sketch and display the number of constraints that are remaining to fully constraining the sketch. This command is also used to solve the sketch in case you have added an additional entity to the already profiled sketch.

AMRULE

This command is used to create a surface using two base wires.

AMSCALE

This command is used to scale the selected surface or wires individually or uniformly in the direction of their axes.

AMSECTION

This command is used to cut one or more section in a selected set of surfaces.

AMSELROT

This command is used to specify a user define rotation point for all the visible objects.

AMSHELL

This command is used to create hollow designer model by defining some wall thickness and removing the remaining material. You can define uniform wall thickness for the designer model or multiple wall thickness.

AMSHOWACT

This command is used to highlight the active designer part or the current sketch plane in the current file.

AMSHOWCON

This command is used to display all the constraints that are applied on the selected profiled sketch in the **Model** mode.

AMSHOWINST

This command is used to highlight all the instances of the selected designer model in the desktop browser.

AMSHOWSKETCH

This command is used to highlight all the geometries on the current sketch plane.

AMSKPLN

This command is used to create a new sketch plane on the specified planar face for creating the features in other planes.

AMSOLCUT

This command is used to cut an AutoCAD solid using the AutoSurf surface that is extending beyond all the faces of the AutoCAD solid.

AMSPLINE

This command is used to create a spline that

is tangent to a specified surface, designer part or an AutoCAD solid.

AMSPLINEDIT

This command is used to edit the control point or the fit point spline.

AMSPLITLINE

This command is used to create a parametric split line that is used to split the faces or the parts.

AMSTITCH

This command is used to stitch the selected surfaces or create a blended surface to fill the gap between the selected surfaces.

AMSTLOUT

This command is used to copy out the selected solids, regions, designer parts or subassemblies in the .STL format to be used for Stereo Lithography.

AMSTYLEI

This command is used to copy only the dimension style from a specified drawing without actually inserting that particular drawing.

AMSUPPRCOLOR

This command is used to control the color of the dimensions and the Degree Of Freedom symbols on the suppressed features.

AMSUPPRESS

This command is used when you do not want some components to be sectioned in the section view created using the scenes.

AMSUPPRESSFEAT

This command is used to suppress the selected feature on the active designer model. This command also allows you to suppress the features with the help of a spreadsheet.

AMSURFCUT

This command is used to cut a selected

designer model using an AutoSurf surface extending beyond all the faces of the designer model.

AMSURFPROP

This command is used to calculate the mass properties of the selected surface to be used during the analysis of the surface model characteristics.

AMSURFSYM

This command is used in the **Drawing** mode to add the surface texture symbols to the drawing view entities. The standards that can be used for the surface texture symbols are ANSI, BSI, CSN, DIN, GB, ISO, and JIS.

AMSWEEP

This command is used to create a complex designer model by sweeping a profiled sketch about a parametric path created using the **AM2DPATH** or the **AM3DPATH** commands.

AMSWEEPSF

This command is used to create a complex surface by sweeping one or more base curves about one or more rails or one augmented line and one rail.

AMSYMLEADER

This command is used to attach or detach a leader from a selected symbol in the drawing mode.

AMSYMSTD

This command is used to select a drafting standard for the BOMs, Balloons, or other symbols used in the drawing views.

AMTEMPLATE

This command is used to modify an existing or create a new holenote template. The new templates are created using the existing standard holenote templates.

AMTEXTSK

This command is used to used to create an

embedded or an engraved text on the designer part. The text material written using this command can be extruded, revolved or swept using the Mechanical Desktop commands.

AMTHICKEN

This command is used to add thickness to the selected AutoSURF surface. The surface is thus converted into a solid part.

AMTOLCONDITION

This command is used in the **Drawing** mode to control the tolerances used by the parametric dimensions.

AMTRAIL

This command is used to add the parametric lines to the exploded or tweaked assemblies displaying the path and the direction of the mating components.

AMTUBE

This command is used to create tubular surfaces along a specified wire that acts as the axis for the tubular surface.

AMTWEAK

This command is used to redefine the position of the assembled components in the **Scene** mode by moving or rotating them.

AMUCSFACE

This command is used to align the UCS to a specified face.

AMUCSPERP

This command is used to define a UCS perpendicular to the selected line. You can also define a sketch plane on the new UCS using this command.

AMUNSPLINE

This command is used to break the selected spline into a polylines fit into a specific tolerance or into fit points.

AMUNSUPPRESSFEAT

This command is used to unsuppress the features that were manually suppressed using the **AMSUPPRESSFEAT** command.

AMUPDATE

This command is used to view the effect of any kind of modifications made to the designer part, assembly, scene or drawing view.

AMVARS

This command is used to create the design variables that can be a numeric value, a parameter or an equation to manage the designer models in the drawing. You can create two types of design variables using this command; the Active Part and the Global design variables.

AMVIEW

This command is used to control the display of the designer models in the **Model** mode. You can rotate the view orientation using this command.

AMVIEWOUT

This command is used to export the drawing views of the designer model created in the **Drawing** mode into a new drawing file in the form of 2D AutoCAD entities.

AMVISIBLE

This command is used to control the visibility of the parts, assemblies, scenes, drawings, work features and other geometric objects.

AMVRMLOUT

This command is used to export the selected object in the form of Virtual Reality Modeling Language so that it can be displayed on the Web pages.

AMWELDSYM

This command is used to create the welding symbols to be added to the drawing views in the **Drawing** mode. The standards that can be used for the welding symbols are ANSI, BSI, CSN, DIN, GB, ISO, and JIS.

AMWHEREUSED

This command is used to find out the location of the selected designer model or the component of the assembly in the current drawing.

AMWORKAXIS

This command is used to create a parametric axis for the circular or cylindrical features in the active designer model. You can also sketch the work axis on the current sketch plane.

AMWORKPLN

This command is used to place a parametric work plane, with the help of the modifiers, for creating the features on the other faces of the designer model.

AMWORKPT

This command is used to place a parametric point at the specified location on the current sketch plane.

AMXFACTOR

This command is used to specify the explosion factor for the scene or individually for different parts of the assembly.

AMXREFCONVERT

This command is used to remove the \$O\$ symbol from the layers, linetypes, blocks, dimension styles, text style and so on of the external references in the current file.

AMZOOM

This command is used to zoom in on to a specified designer part or subassembly.