RECSOFT Inc.

EasyDone for AutoCAD

U SER ' S G UIDE

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Product Overview

Raster-to-vector products, based on automatic text and graphics recognition, are used in different CAD systems for last 15-20 years, however they could not become very demanding systems, at least as it was expected in late 80s. There are several reasons for that, but the main one is that automatic recognition cannot provide, in general, high reliability of recognition results, required for the most of CAD tasks. Many of paper drawings have not a very good quality and were not prepared with the idea of their input by scanner and following automatic recognition, so, if some raster-to-vector programs show reliable enough result on specially prepared samples (usually with fine quality), they cannot show the same reliability on real documents. Recognized graphical elements often have too many structural distortions and recognized text contains too many errors.

In this connection, users prefer do not mess with automatic recognition at all but, instead, convert documents in vector format usual way.

Product EasyDone is not free of all of these disadvantages, of course, but their negative effect is reduced to the possible minimum.

Automatic graphics recognition works so, that as large as small graphical objects are recognized with minimal distortions. Graphics recognition provides acceptable results for a wide range of drawing types, may be just except too complex mechanical drawings, which have a lot of hatched areas. It works reliable even on documents with poor quality, and does not require any special tuning.

Automatic text recognition does not require any training period and provides quite reliable results on documents of good and average quality. For documents with poor quality EasyDone has special mode called "Semi-automatic text recognition". This mode is an original RECSOFT's new technology of text input and is convenient for using as alternative of fully automatic text recognition. Automatic text recognition practically always requires the following step of checking of recognized words, but semi-automatic recognition does not have this disadvantage. In this connection some customers may prefer to use semi-automatic recognition even on documents of good quality rather than check all recognized words.

Checking of recognition result is a common problem of all automatic raster-to-vector systems, but not many of products place high emphasis on this thing and prefer to leave it to the native CAD editing tools. However, such CAD tools are designed most likely to create some new objects, so they are not effective and convenient for correction of results of automatic recognition.

To solve this problem, EasyDone is equipped with special built-in Recognition Result Editor, which is maximum effective and convenient exactly in editing of recognition results.

RECSOFT Inc. hopes that you will enjoy this product, which will be really useful in your work.

Open/Save Recognition Parameter Set

EasyDone performs recognition of text and graphics objects according to the current setting of text and graphics recognition parameters. This setting defines what type of entities should be recognized (for instance polylines, circles, solids, text strings, etc.), which layers should be assigned to each type of recognized entities, how to recognize some particular types of entities (for instance with or without width filtration) and many other important factors.

Current setting may be defined by direct changing of text and graphics recognition parameters in the corresponding EasyDone dialog boxes, but may be also loaded from previously saved or pre-defined file, such as for example file "Default.rps", which is supplied with EasyDone and automatically placed in the folder <Parameter Files> in the EasyDone working directory at the moment of installation. You may use this file as a prototype for your own parameter settings because it is easier to change just a few pre-defined parameters according to your current needs then to set up all parameters from the scratch.

Note that parameters, which are subject for saving, do not include parameters of Batch Recognition and parameters of Recognition Result Editor.

Starting Recognition

Before you can start recognition or raster editing with EasyDone, it is necessary to attach to the drawing a raster image. EasyDone does not have its own commands for raster attaching (to avoid confusion with similar AutoCAD commands), so you should attach required raster image by AutoCAD menu "Insert/Raster Image".

EasyDone provides three recognition modes invoked from EasyDone menu or toolbar:

- Recognize Text+Graphics
- Recognize Graphics
- Recognize Text

In addition to these modes EasyDone provides batch recognition, when user first creates a list of documents to be recognized (menu item "Batch Recognition\Set Up Batch Recognition") and then starts batch recognition (menu item "Batch Recognition\Start Batch Recognition").
After that EasyDone automatically recognizes all documents on the batch list one by one and places the output DWG files according to the parameter "Output Path", defined in the "Set Up Batch Recognition" dialog box.

The first step in starting of recognition process in regular (not batch) mode is defining of the zone, which should be recognized.

The following EasyDone command appears in the command line:

[Define Recognition Zone]New/Previous/Esc/<Whole_image>:

This command allows selection of the following options:

Option:	Mnemonic:	Description:
Whole_image	W	Recognition of the whole document
Previous	P	Recognition in the previous zone (if it was defined earlier)
New	N	Define new zone for recognition within current document
Esc	E	Exit from the selected recognition mode

The default selection is **Whole_image**, so you need just to press Enter or right mouse button for this selection.

If you select option **New**, the following EasyDone command appears in the command line:

[RECTANGLE]Polygon/Corner #1

or

[POLYGON]Rectangle/First point

The **[RECTANGLE]** command allows to define rectangular recognition zone or to switch to polygonal zone selection (mnemonic **P**).
The **[POLYGON]** command allows to define polygonal recognition zone or to switch to rectangular zone selection (mnemonic **R**).

Note: Areas of 16 pixels width cannot be recognized on the edges of image because of algorithm requirements.

TIPS:
You may cancel definition of recognition zone as well as the recognition process itself by pressing Esc.
You do not have to close polygonal zone - after the last point of the polygonal zone is selected and Enter or right mouse button is pressed, the polygon is closed automatically.

After selection of recognition zone is done, EasyDone begins automatic recognition of the document according to the current setting of text and graphics recognition parameters.

Batch Recognition

Batch Recognition is a convenient option for recognition of a sequence of raster files directly after input by scanner. No additional operations on these files are required for batch recognition.
You may use batch recognition also for previously saved DWG files, which have attached raster image (note that each DWG file should have only **one** attached image).

Before starting of batch recognition it is necessary to set up some common settings and create Batch List.
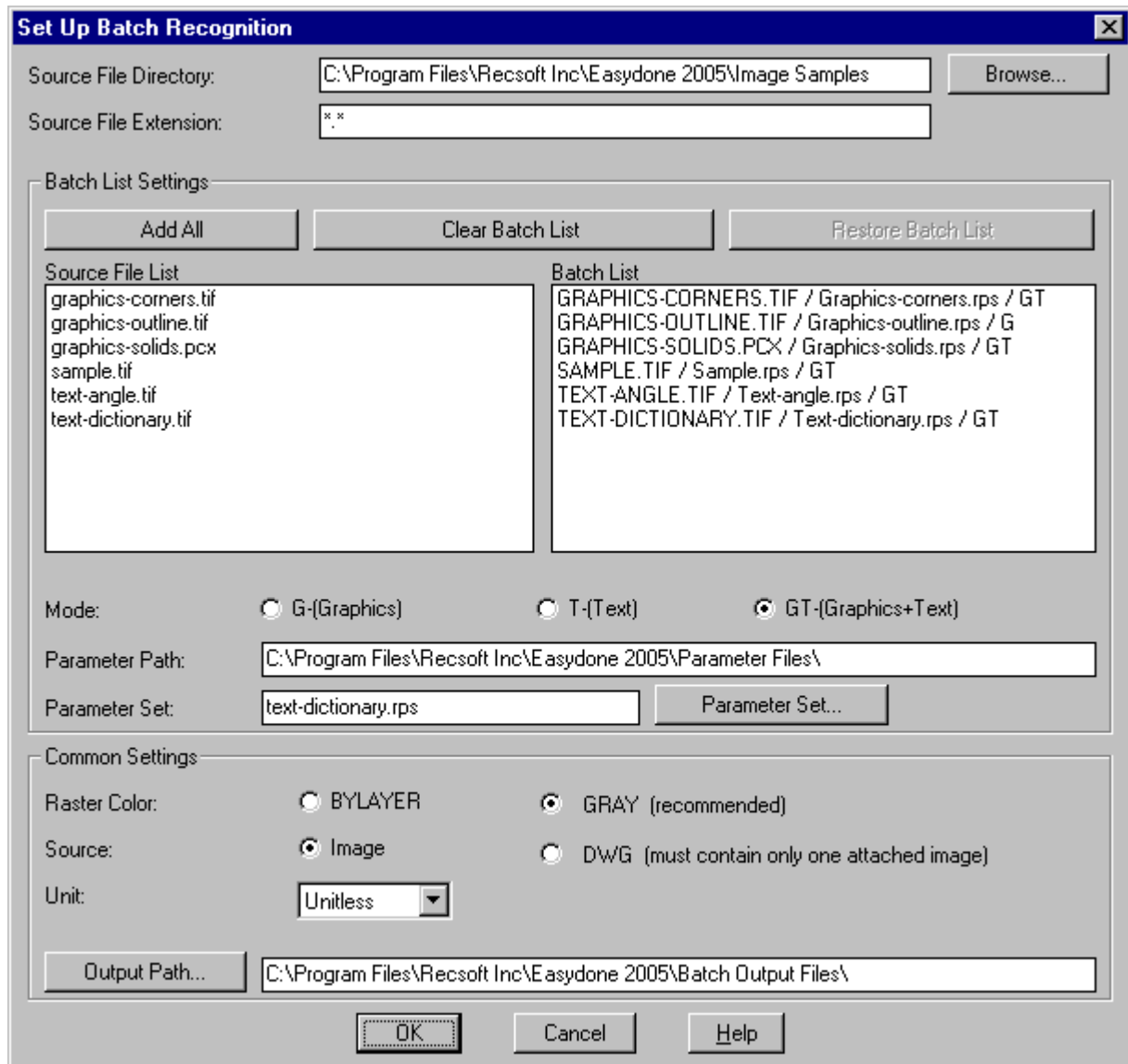
Common settings for batch recognition are the following:
- Path to Source Files;
- Path to Parameter Set Files;
- Path to Output Files;
- Raster Color (BYLAYER or Gray);
- Source Type (Image or DWG);
- Unit.

Path to Source Files provides access to the folder where raster or DWG files for batch recognition are located. You may use button "Browse" to find required path.

Path to Parameter Set Files provides access to the folder where parameter files for batch recognition are located. You may use button "Parameter Set" to find required path.

Path to Output Files provides access to the folder where Batch Recognition will save all output DWG files and file <batchlog.txt> with batch recognition report (also in some cases file <batchwrn.txt> with error and warning messages). You may use button "Output Path" to find required path.

Raster Color parameter defines what color to use for images, attached to output DWG files.
This parameter is available only for "Image" source type because for DWG source type color is already defined in each DWG file.

Source Type parameter defines whether recognized documents are raster images or DWG files.

Unit parameter defines what unit type to use for output DWG files.
This parameter is available only for "Image" source type because for DWG source type unit is already defined in each DWG file.

Note: When you change Path to Source Files, Path to Parameter Set Files or Source Type, the previously created Batch List will be cleared and its creation will start from the beginning (if you want to undo deleting of the previous Batch List, you may leave dialog box by button Cancel).

After all common parameters are defined it is necessary to create Batch List using Source List.
Source List appears automatically according to the selected source folder and current list in the **Source File Extension** field.
If you want to place on the Source List all files from the source folder, you may type ***.*** in that field or simply clear the field by mouse or Delete key and then press Enter.
If you want to place files with some particular extensions (for instance TIF, PCX, etc.), you may type these extensions in the Source File Extension field separated by commas or spaces in the format ***.EXT** (for instance: *.tif, *.pcx). You may also use button "Browse", which allows not only to select source folder, but also select required extension (by clicking any file with desired extension in the source folder). If this extension is not on the list yet, it will be automatically added.

The process of creation of Batch List consists of three operations, in general case.
1) Select recognition **Mode** (Graphics, Text or Graphics+Text);
2) Select **Parameter Set**;
3) Click source file on the Source List.
(If current value of Mode or Parameter Set is OK, then you need only to click source file.)
After you click source file on the Source List, the new line appears on the Batch List. It contains: name of the source file; name of the parameter file, which will be used for recognition of this source file; recognition mode.
Then you should repeat all these operations until Batch List is complete.

If you want to delete some line from the Batch List, you may click it by mouse and the "Batch List Item" panel will appear. You may use this panel to delete item or to see the name of source file in wide format, if this name is too long to fit in the window with Batch List.

In the case when you want to place on the Batch List all files from the Source List with the same values of Mode and Parameter Set, you may use button "Add All".

There are also auxiliary buttons "Clear Batch List" and "Restore Batch List", which allow to remove all items on the Batch List at once or to restore them.

After Batch List is complete, you may quit "Set Up Batch Recognition" dialog box (press button OK) and use menu "EasyDone\Batch Recognition\Start Batch Recognition" to begin batch recognition process.

Batch Recognition Notes and Tips:

1. To avoid unpredictable condition of AutoCAD environment, that may affect proper work of batch recognition, it is highly recommended to reload AutoCAD before starting of batch recognition (if you just created Batch List, reloading is not required).

2. If batch recognition does not start, take a look at your AutoCAD text screen - it may help to figure out the reason of wrong situation. If batch recognition did not start at all or failed without any messages, check that AutoCAD is started only once. Windows is multi-task system and, if more than one AutoCAD is currently started, it may affect normal work of Batch Recognition. It is not recommended also to start

Batch Recognition while other than AutoCAD programs are working in parallel, especially if they require significant amounts of memory or processor resources.

3. If batch recognition has failed, you need to read the warning file and batch recognition report, which are located in the same directory as output DWG files.

If there are no records about the reason of batch recognition failure, you should start recognition of the failed source file in the standard mode (don't forget to load the same parameter set before).

In the standard mode you will receive error messages on the screen, so it may help to understand the reason of failure.

Assigning Different Fonts

Each recognized text entity is assigned to the text style defined in Text Recognition Parameters dialog box for the associated class. If you want to assign to recognized text some particular font, you may do it by changing corresponding font selection in the AutoCAD dialog box "Text Style".

Let we assume that there are two classes defined in Text Recognition Parameters dialog box.
The first class with yellow color and text style "Style1", the second class with blue color and text style "Style2" (that is setting from the tutorial parameter set SAMPLE.RPS).

If default AutoCAD font (usually "txt.shx") was not changed, then recognized image will look like on the illustration below.
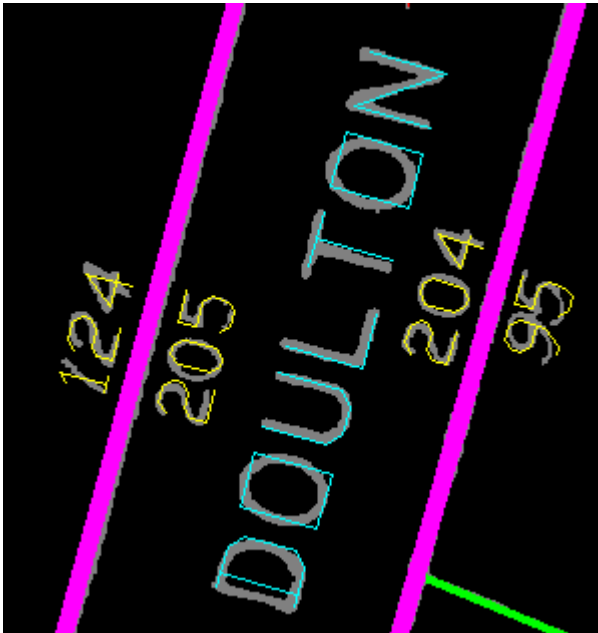
As a sample we will select for the first text style font "Kino MT" (to open AutoCAD dialog box "Text Style" you may type command STYLE or use AutoCAD menu "Format\Text Style").
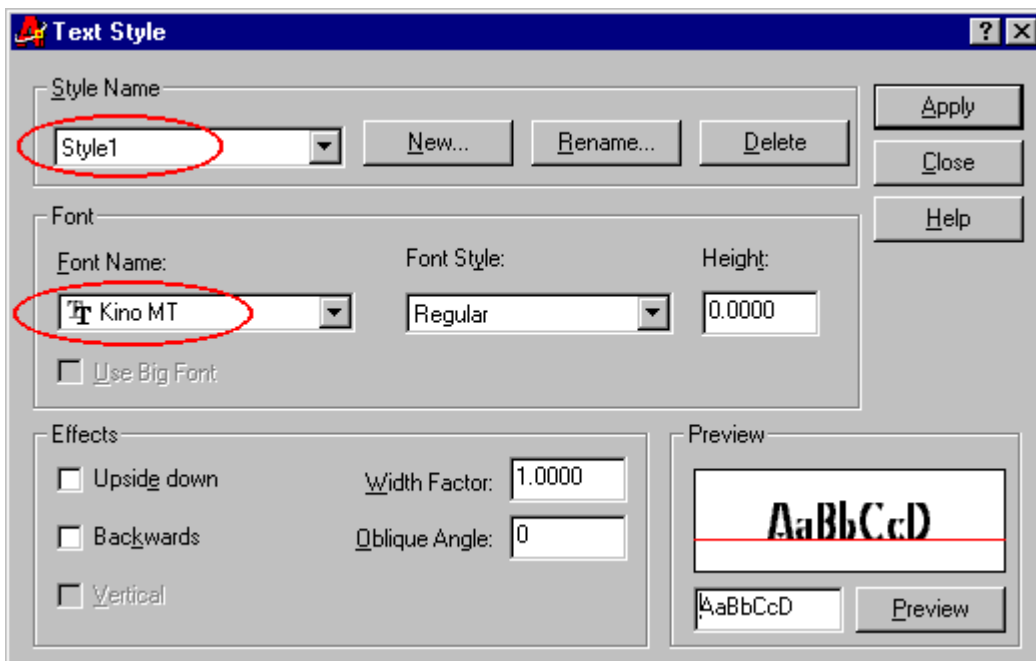
Then we will select for the second text style font "Desdemona".
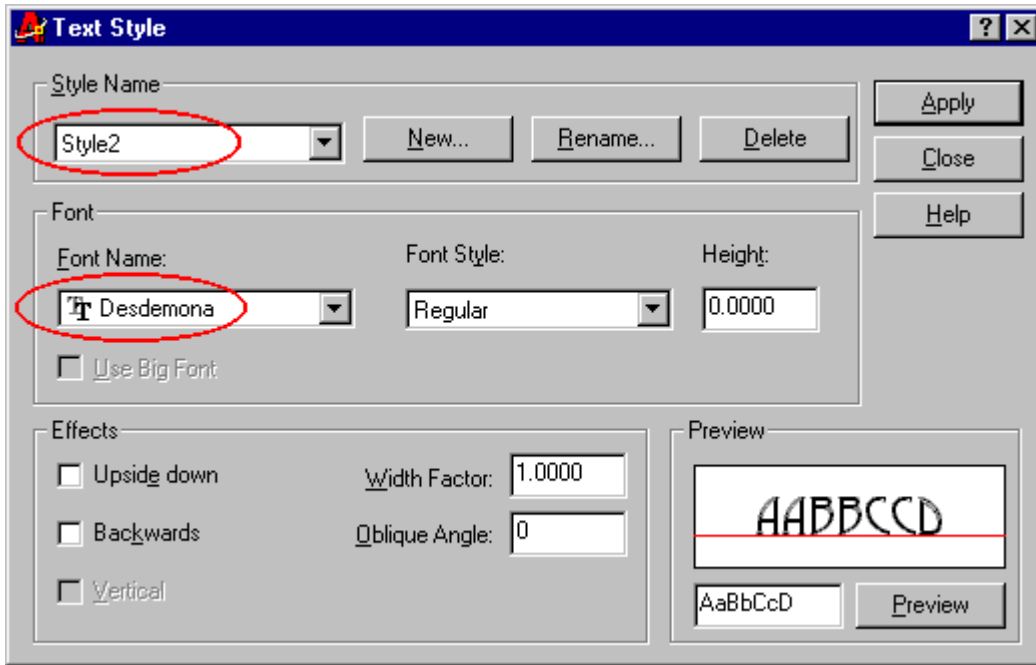
After that the recognized image will look the following way.
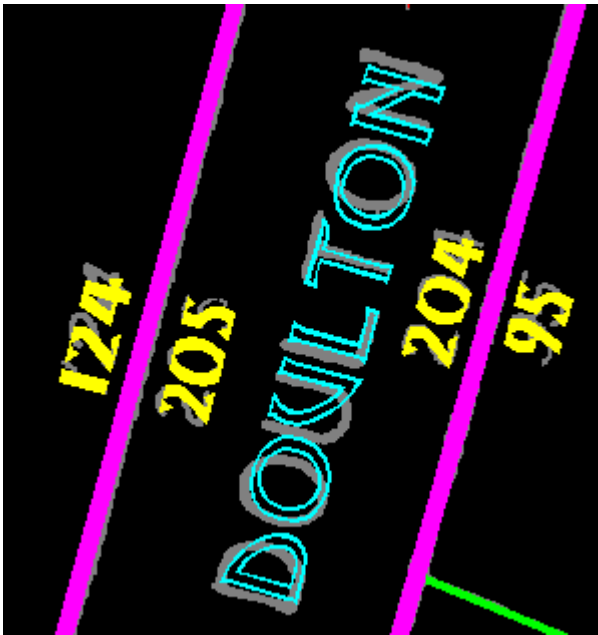
Recognition of non-English Characters

EasyDone provides recognition of characters of the standard "Latin-1" character set, so it may be used for recognition of texts in different languages (French, German, Italian, Spanish, etc.). By default (after installation) recognition of non-English characters is turned off. To turn recognition of required characters ON, you should open dialog box "International Character Set" (from dialog box "Text Recognition Parameters") and check necessary characters.
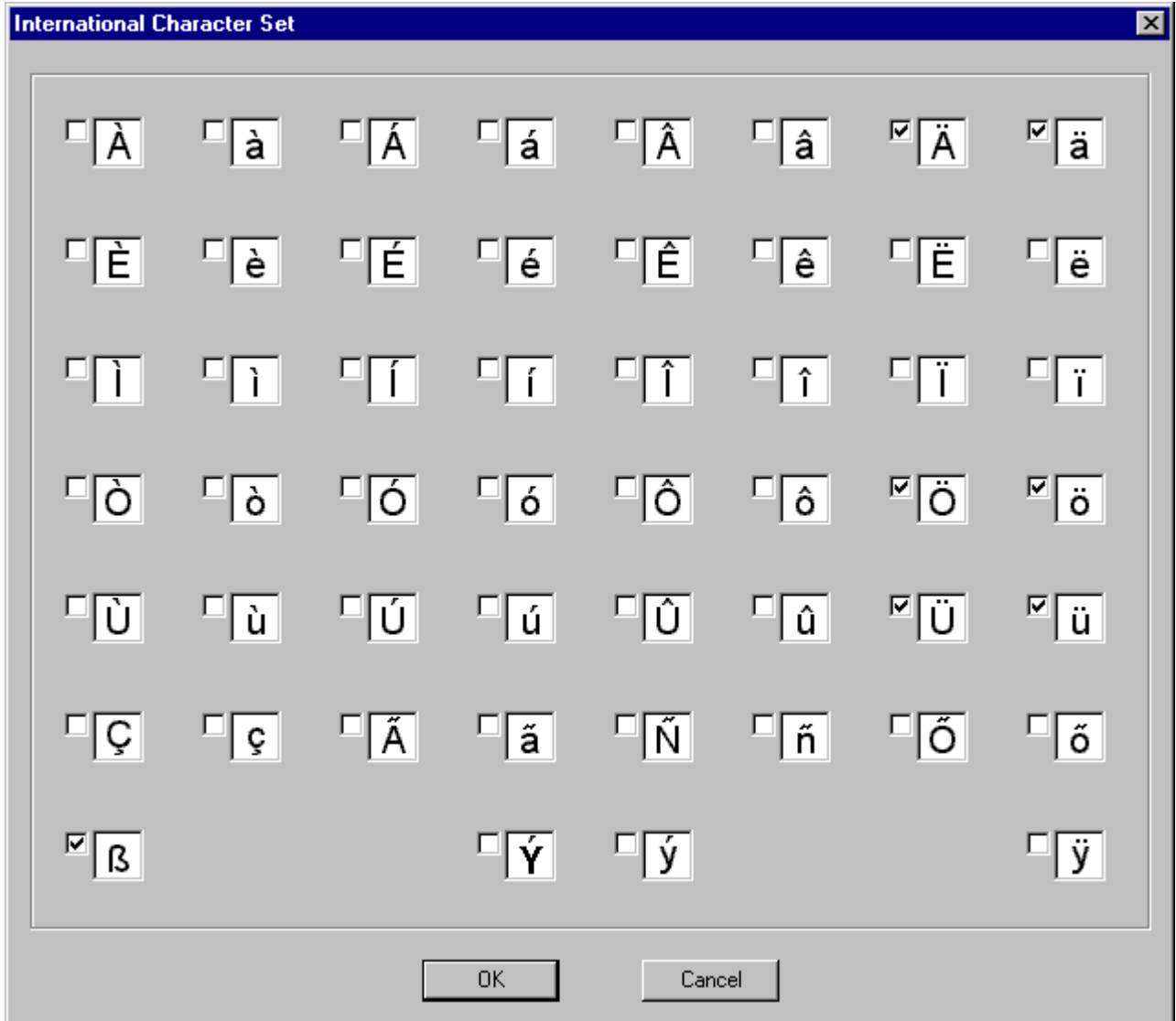
Sample of turning on recognition of German character set

Note that EasyDone can recognize also many characters of extended Latin set, which is used in some Eastern European languages, however recognition of these characters is not included in the standard configuration of the product. If you are interested in recognition of such characters, please contact our technical support on this matter.

Graphics Recognition Parameters

Default Type

The Default Type option sets up pre-defined values to some topological graphics parameters, such as Straighten Lines, Adjust to Orthogonal, Ignore Noise and others.

Contour map
Choose this selection to set parameters that recognize graphics without any line correction.
It is recommended for GIS (contour maps, soil maps, etc.) or similar drawings where polylines must be as close to raster lines as possible.

General GIS
Choose this selection to set parameters that recognize graphics with minimal line correction.
It is recommended for GIS drawings like parcel maps or similar drawings where polylines must be close to raster, but should not have too much vertices.

Architect./Mechan.
Choose this option to set parameters to recognize precision graphics with mostly straight orthogonal lines and small details, such as floor plans or mechanical drawings.

Schematic
Choose this option to set parameters to recognize graphics with significant line correction.
Electrical P&ID diagrams are an example of a drawing that would use the schematic option.

Diagram
Choose this option to set parameters to recognize graphics with strong line correction.
Flow block diagrams are an example of a drawing that would use this option.

Table
Choose the Table option to set parameters that recognize manually drawn tabular graphics by automatically correcting non-orthogonal lines, over shoots, and under shoots.
Some examples of drawings that would use the table option include mechanical charts, furniture or door schedules, and process and instrumentation drawings.

Straighten Lines
This parameter sets the maximum deviation of a particular vector from the corresponding raster line.
The vector is straightened to compose one straight line-segment.
0 - vector closest to raster
7 - yields straightest vector
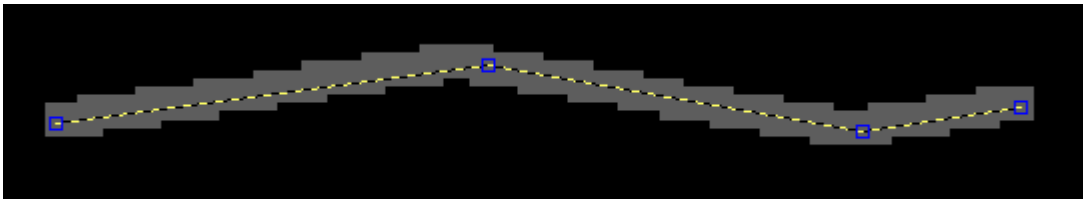
Result of recognition with parameter value "0"

Result of recognition with parameter value "7"

Adjust to Orthogonal
This option sets the parameters to rotate lines that are close to orthogonal, (e.g. vertical and horizontal), to be strictly orthogonal.

Ignore Noise
This option allows user to define the maximum pixel length of lines that are to be deleted as background noise.

TIP: Parameters Straighten Lines, Adjust to Orthogonal and Ignore Noise are defined automatically according to selected Default Type. You may use them for fine-tuning, however it is rare really necessary.

Layer/Color
This option allows user to save graphics primitives to a specified layer. You may type the layer name in the edit field or press button **[>>]** to select layer/color from other dialog boxes or current drawing.

Note: If the Size Filters and/or Width Filters options are selected to filter graphics, the Layer and Color options will be disabled. To define the Layer and Color options use the specific dialog boxes.

Polyline Recognition Options
In this dialog box you may set up desired mode for recognition of polylines or single segments.

Single segments only (break at intersections)
This selection creates straight polyline segments with two vertices only.
The segments created will be broken and have end points at the intersections.

Single segments only (pass intersections)
This selection creates straight polyline multi-vertices segments with automatic passing of intersections.

Break polyline at all intersections
The polylines created will be broken at the intersections.

Follow continuous curves
The polylines created will not be broken and will have vertices in intersections.

This setting is commonly used for wide range of drawings, such as: contour and parcel maps, mechanical and architectural drawings, etc.

Size Filters

This selection allows user to enable the Size Filters to save small isolated raster graphics such as symbols and text to a separate layer. When turned ON, this variable disables the Layer and Color option in this dialog box. The Layer and Color will be set in the Size Filters dialog.
The "Size Filters" button calls the corresponding dialog box.

Width Filters

This selection allows user to enable the Width Filters to save raster lines with different widths to separate layers. When turned ON, this variable disables the Layer and Color option in this dialog box. The Layer and Color will be set in the Width Filters dialog.
The "Width Filters" button calls the corresponding dialog box.

Circles

This selection allows user to enable recognition of circles and to set up desired parameters.
The "Circles" button calls the corresponding dialog box.

Arcs

This selection allows user to enable recognition of arcs and to set up desired parameters.
The "Arcs" button calls the corresponding dialog box.

Solids

This selection allows user to enable recognition of solids and to set up desired parameters.
The "Solids" button calls the corresponding dialog box.

Dash

This selection allows user to enable separation of dash lines and to set up desired parameters.
The "Dash" button calls the corresponding dialog box.

Hatch

This selection allows user to enable separation of hatch lines and to set up desired parameters.
The "Hatch" button calls the corresponding dialog box.

Arrows

This selection allows user to enable recognition of arrowheads and to set up desired parameters.
The "Arrows" button calls the corresponding dialog box.

Outline

This dialog box allows user to set up the desired parameters for raster outlining.
The "Outline" button calls the corresponding dialog box.
Note: If the Outline mode is enabled, all other graphics recognition modes are disabled.

Measure button
This command is used to measure height, width, and associated angle of different raster objects.
The measured values in pixels and World Coordinates (WC) will appear on the bottom line of the current dialog box. These values may be required to set up some graphics and text parameters.
See the topic "Additional Operations/Menu "Measure Size" for information how to use command.

Polyline Recognition Options
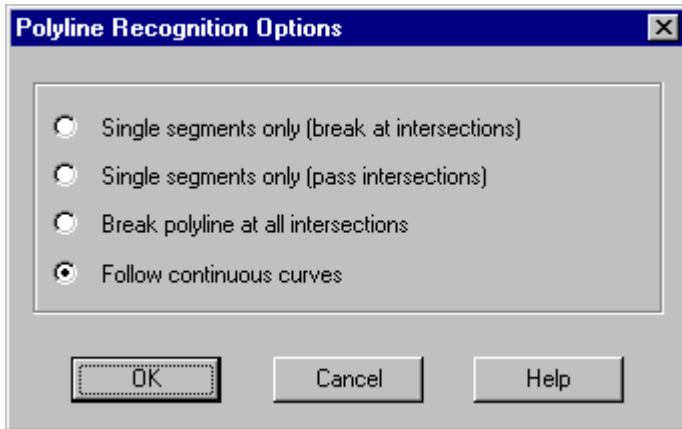
Single segments only (break at intersections)
This selection creates straight polyline segments during recognition.
The segments created will be broken and have end points at the intersections.
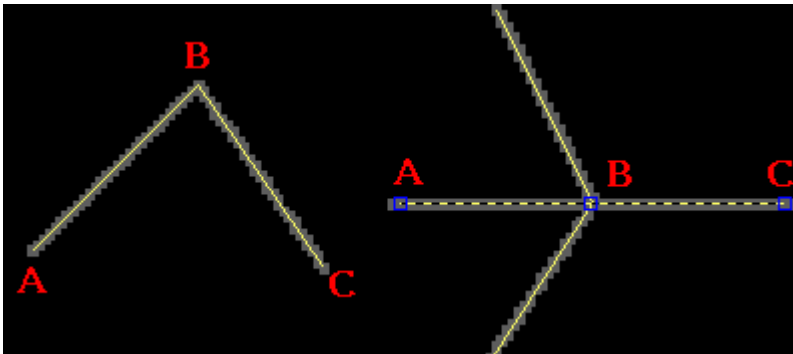
A–B is one polyline,   B–C is another polyline

Single segments only (pass intersections)
This selection creates straight polyline segments during recognition.
These segments will not be broken at the intersections.
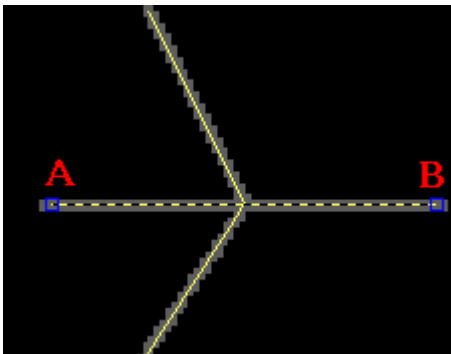
A–B is one polyline

The polylines created will be broken at the intersections.
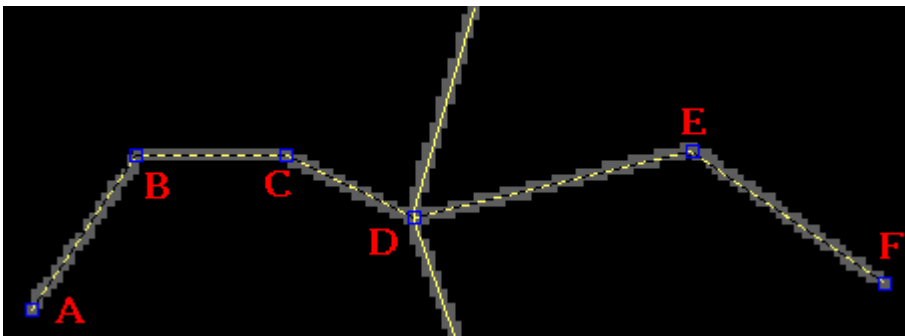
A–B–C–D is one polyline,   D–E–F is another polyline

The polylines created will not be broken and will have vertices in intersections.
This setting is commonly used for wide range of drawings, such as: contour and parcel maps, mechanical and architectural drawings, etc.
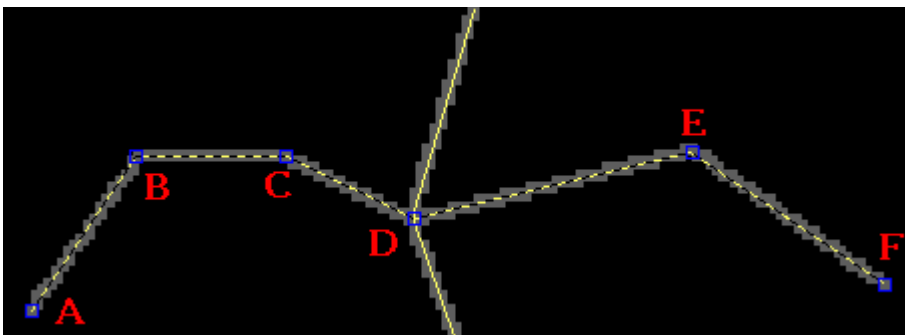
A–B–C–D–E–F is one polyline

Graphics Size Filter Parameters
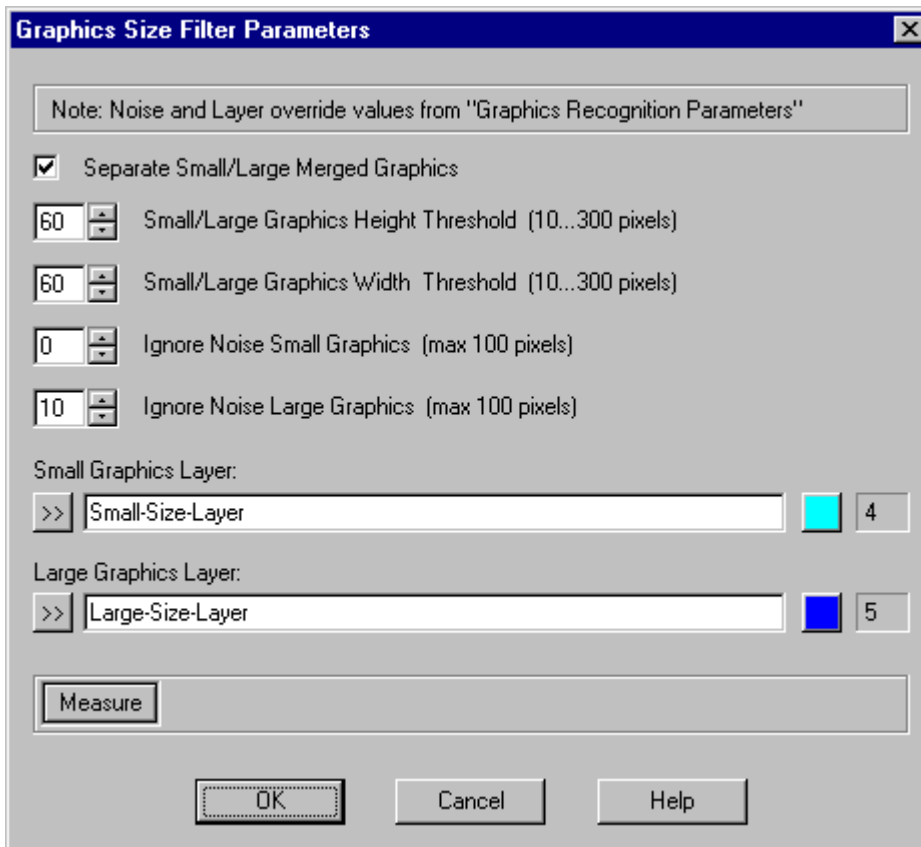
The Graphics Size Filter Parameters dialog box sets the parameters to classify general objects according to their height and width. These classifications are then split into two groups: Small Graphics and Large Graphics.

The graphical objects with either height or width exceeded by the permissible value are assigned to the Large Graphics layer.

All others are assigned to the Small Graphics layer.

Size Filtration may be useful in many cases, for instance if you do not want to plot text but only graphics. In this case you may use recognition of graphics only and assign small graphics (that is much likely text) to a separate layer and than freeze this layer or mark it as "Do not plot".

Size Filtration may be used also to assign graphical symbols to a separate layer, and in other similar cases.

Separate Small/Large Merged Graphics

Set this option ON in order to separate into different layers small and large merged objects.

See illustration below ("Large" layer has magenta color and "Small" layer has green color).

Result with parameter turned OFF

Result with parameter turned ON

Note: Height and Width Thresholds should be set up higher than actual size of "small" objects (text in this sample) otherwise separation is not possible.

Small/Large Graphics Height Threshold
This sets the permissible height of the graphical objects to be considered as "small".
The range of allowable values is 10 to 300 pixels.

Small/Large Width Threshold
This sets the permissible width of the graphical objects to be considered as "small".
The range of allowable values is 10 to 300 pixels.

Ignore Noise Small Graphics
With Size Filtration set to ON, the "noise threshold" is different for large and small graphics.
This variable sets the noise removal threshold for the small graphics objects.
Any "noise" smaller than the pixel value entered is ignored, and no vector is created.

Ignore Noise Large Graphics
This variable sets the noise removal threshold for the large graphics objects.
Any "noise" smaller than the pixel value entered is ignored, and no vector is created.

Small Graphics Layer
This option sets the layer/color setting for all the graphical output objects included in the Small Graphics Layer.

Large Graphics Layer
This option sets the layer/color setting for all the graphical output objects included in the Large Graphics Layer.

Graphics Width Filter Parameters

The Graphics Width Filter Parameters dialog box allows you to save raster lines with different width to eight different layers.

Width Limits
These boxes define the minimum and maximum raster line width in pixels for each class.
Note: When you set up parameter "Width Limits", it is recommended to define minimum and maximum width for each class so, that average value of this parameter for each class differs from average value in other classes at least for 3-4 pixels. The point is that actual width of lines on real raster has a lot of fluctuations and, if difference between classes is too minor, it may result in wrong width filtration.

Output WC
These boxes define the output line width in World Coordinates (WC). You may measure this value (button "Measure") for raster lines of each class and then set it up in these boxes.
If you place "-1" instead of output width, lines will have output width, which is counted as an average value of minimum and maximum pixel Width Limits converted into WC.

Note (except mode "Use actual width of polyline segments"):
You should understand that if you set up "-1" for some class then width of recognized line will not have any relation to actual width of this line on raster and will be controlled only by parameter Width Limits for this class. For instance, if you defined Width Limits for this class as 2-6 pixels, then output line will have width like 4 pixels. If Width Limits are defined as 2-10 pixels, then output line will have width like 6 pixels.

So, it is recommended to define Width Limits as more precisely as possible if you would like to use automatic width counting (value "-1"). See illustration below.

Original raster line (width 10 pixels)

Width Limits 8-10 pixels (average 9 pixels) and Output WC = -1

Width Limits 8-22 pixels (average 15 pixels) and Output WC = -1

The following options define how to interpret varying line width and width of small objects.

Use average width of polyline segments
Polyline will be assigned to the width class corresponding to the average width of polyline segments.

Use different width of polyline segments
Each segment will be filtered individually and will have its own width adjusted to the corresponding width class. Polyline will be assigned to the class corresponding to segments with maximum summary length.

Use actual width of polyline segments
The layer/color of polyline will not be defined by width class, but each segment will have its own width according to the actual pixel width on raster.
Optional width adjustment
You may use this parameter along with above option for minor width adjustment of all segments.

Break polyline apart if segments have different width
EasyDone actually breaks a polyline when its width goes out of the current class and puts each part of polyline on the corresponding layer.

This setting allows you to recognize "small graphics" (see definition in the topic "Graphics Size Filter Parameters") with width option if parameter is turned <On>, however it is not recommended unless you have to use it for some specific needs.

Note 1: If some straight segment has different width on raster along its length, this difference may be ignored because EasyDone can assign some width value only in the ends of segments (polyline vertices).

Note 2: Width counted on raster is always adjusted to the closest even value. For example, if one raster line has actual width 7 pixels and another line has actual width 8 pixels, they both will be interpreted as lines with raster width 8 pixels.

Note 3: If you use mode "Break polyline apart if segments have different width" and option "Arc_in_Polyline" (see the topic "Arc Recognition Parameters"), it may result in appearing of additional break points on arc-segments.

Circle Recognition Parameters
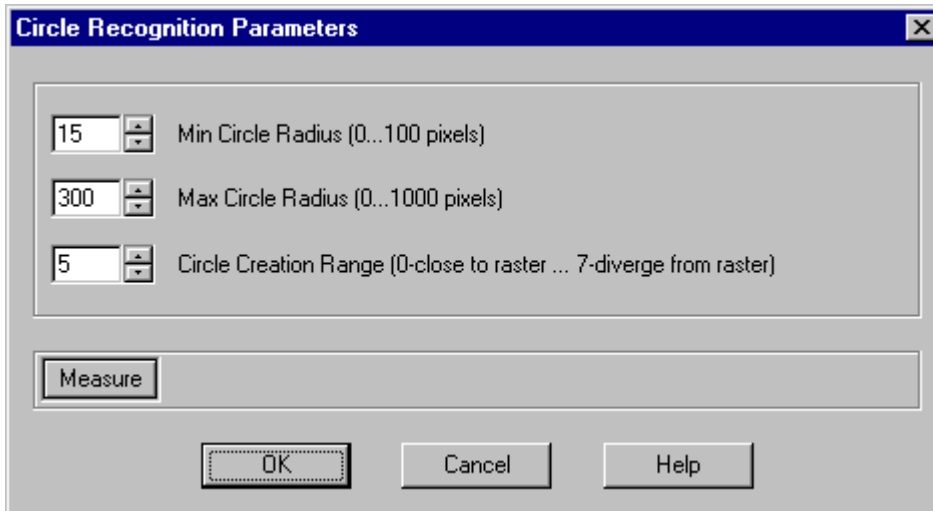
The Circle Recognition Parameters dialog box lets you choose whether EasyDone should convert circular entities. If this feature is not selected, then any circular raster images will be drawn as polylines.

Min Circle Radius
This parameter allows you to select the minimum radius for circular entities being recognized.
Any circles with a radius smaller than the selected value will be recognized as polylines.

Max Circle Radius
This parameter allows you to limit the size of converted circular entities (0 - 1000 pixels).
Any circles with a radius larger than the selected value will be recognized as polylines.

Circle Creation Range
This parameter allows setting up the accuracy of conversion of circular entities in the range from 0 (the highest accuracy, the resulting circle has minimum deviation from raster lines) to 7 (the lowest accuracy, the resulting circle has maximum deviation from raster lines).
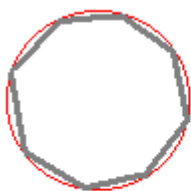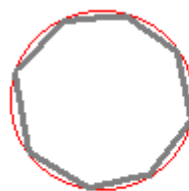
Creation Range = 0   Creation Range = 7

Note: If option "Width Filters" in "Graphics Recognition Parameters" dialog box is turned ON, circles will be recognized as polylines, which consist of one arc-segment closed on itself. If this option is turned OFF or if option "Arc_Entity" is turned ON (see the topic "Arc Recognition Parameters"), then circles will be recognized as AutoCAD CIRCLE entities. Using of CIRCLE entity instead of polyline is required if you convert recognized polylines from LWPOLYLINE to 3D format.

Arc Recognition Parameters
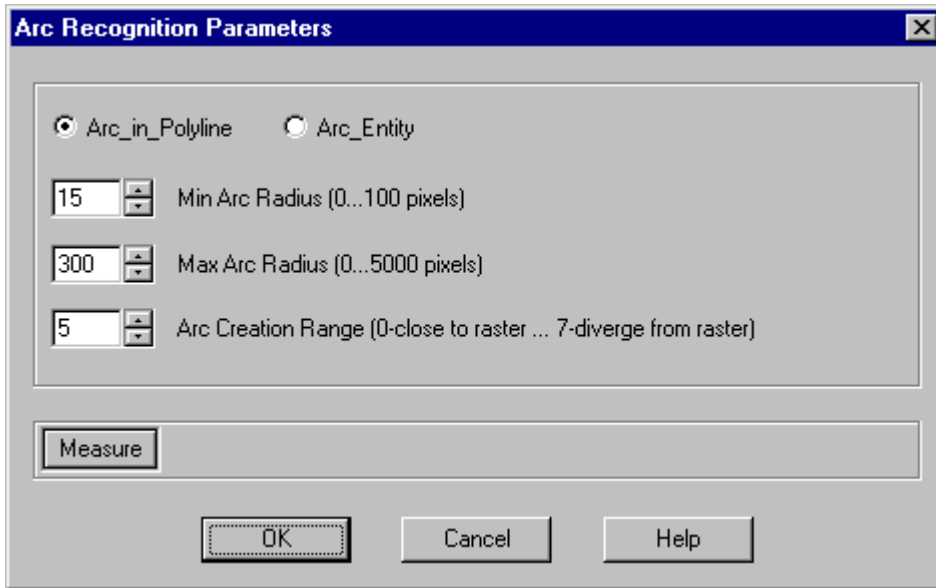
The Arc Recognition Parameters dialog box lets you choose whether EasyDone should create arcs on curved parts of raster lines. If this feature is not selected, then any curved raster line will be drawn as a polyline.

Arc_in_Polyline and Arc_Entity
This selection defines whether arc will be recognized as part of polyline (vertex with bulge) or as AutoCAD ARC entity. If you select Arc_Entity, the created arcs will break polyline apart and will not be filtered by width. Using of Arc_Entity selection is required if you convert recognized polylines from LWPOLYLINE to 3D format.
Note: Arc_Entity selection results also in recognition of circles as AutoCAD CIRCLE entities.

Min Arc Radius
This parameter allows you to select the minimum radius for arcs being recognized.
Any arcs with a radius smaller than the selected value will be recognized as polylines.

Max Arc Radius
This parameter allows you to limit the size of created arcs (0 - 5000 pixels).
Any arcs with a radius larger than the selected value will be recognized as polylines.

Arc Creation Range
This parameter allows setting up the accuracy of creation of arcs in the range from 0
(the highest accuracy, the resulting arc has minimum deviation from raster lines) to 7
(the lowest accuracy, the resulting arc has maximum deviation from raster lines).
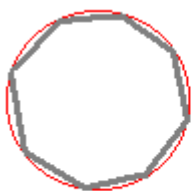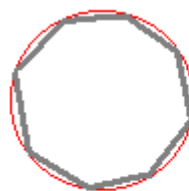
Creation Range = 0   Creation Range = 7
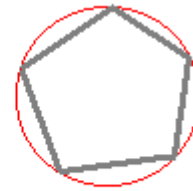
Solid Recognition Parameters
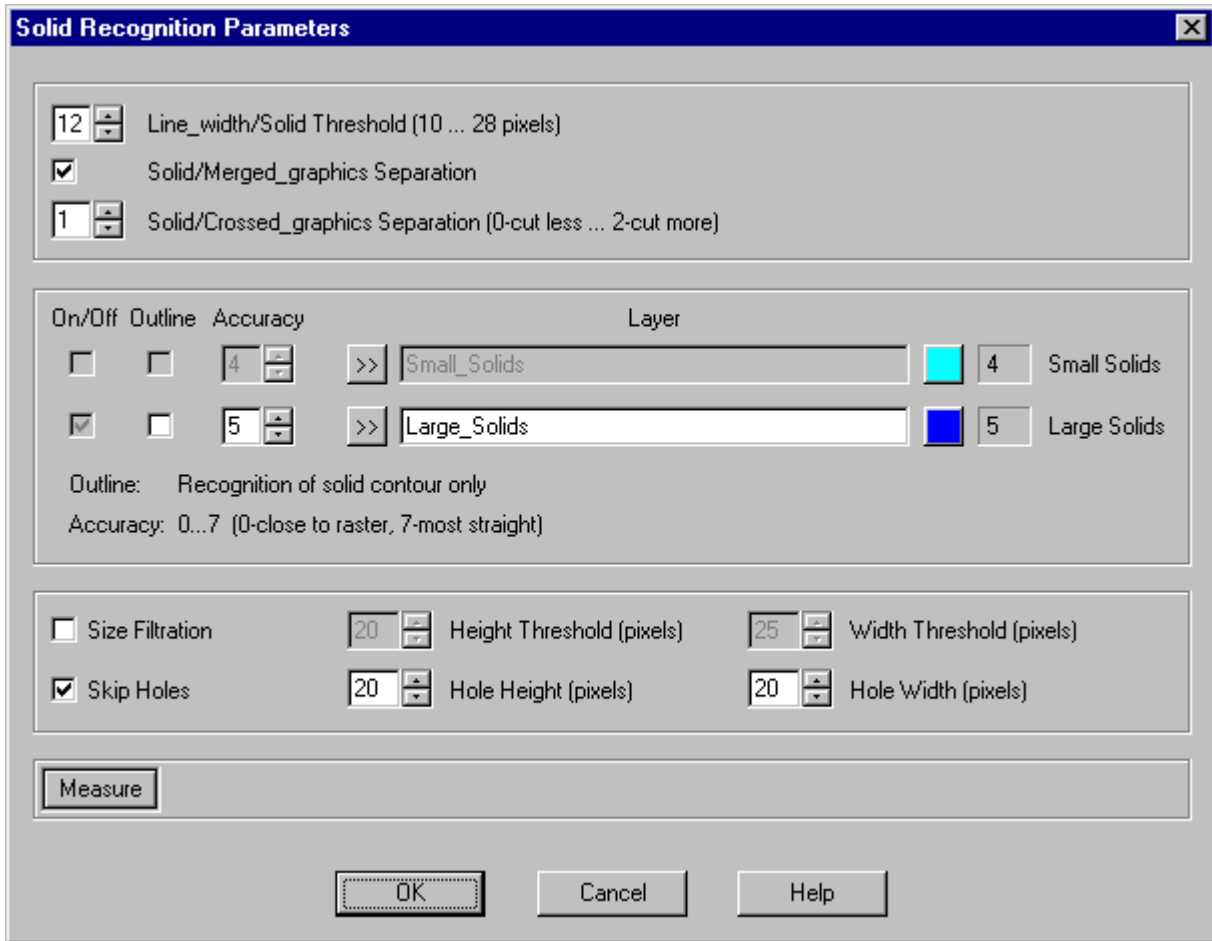
Line_width/Solid Threshold
This parameter defines in which case object should be interpreted as solid or as graphics line.
If length and width of some raster object (line or spot) exceeds the value of this parameter, the object will be recognized as solid, otherwise as graphics line.

Note: If option "Arrow" in "Graphics Recognition Parameters" dialog box is turned ON, EasyDone will decide automatically whether to recognize solid or arrowhead when both entities may be recognized on the same raster spot, but it is not possible to recognize arrowhead on one part of raster spot and solid on the remaining part.

Solid/Merged_graphics Separation
This parameter allows enabling or disabling of automatic separation of solid from merged graphics lines.
The "merged graphics" means graphics line(s), which touch solid from its side(s).
If this parameter is ON, graphics line(s) will be recognized separately from solid as independent object, otherwise the part of line(s) touching solid will be included in the solid body.

Solid/Crossed_graphics Separation
The "crossed graphics" means graphics line(s), which cross the body of solid.
Value "0" ("Cut less") defines that parts of line, intersecting solid contour, should belong to solid.
Value "2" ("Cut more") defines that parts of line, intersecting solid contour, should belong to line.
Value "1" defines intermediate solution.

Size Filtration
This option allows to separate recognized solids to "small" and "large" classes and to set up different parameters (if necessary) for each class.
If this option is turned OFF, all recognized solids are assigned to large class.

If this option is turned ON, the threshold of height and width between small and large classes is defined by parameters "Height Threshold" and "Width Threshold". When both height and width of solid do not exceed corresponding thresholds, solid is assigned to small class, otherwise to large class.

Skip Holes

If solids have some meaningless holes inside their bodies (for instance because of poor quality of scanner input), it is possible to skip such holes.
The height and width of holes, which should be skipped, is defined in parameters "Hole Height" and "Hole Width".

Dash (Dot) Line Separation Parameters
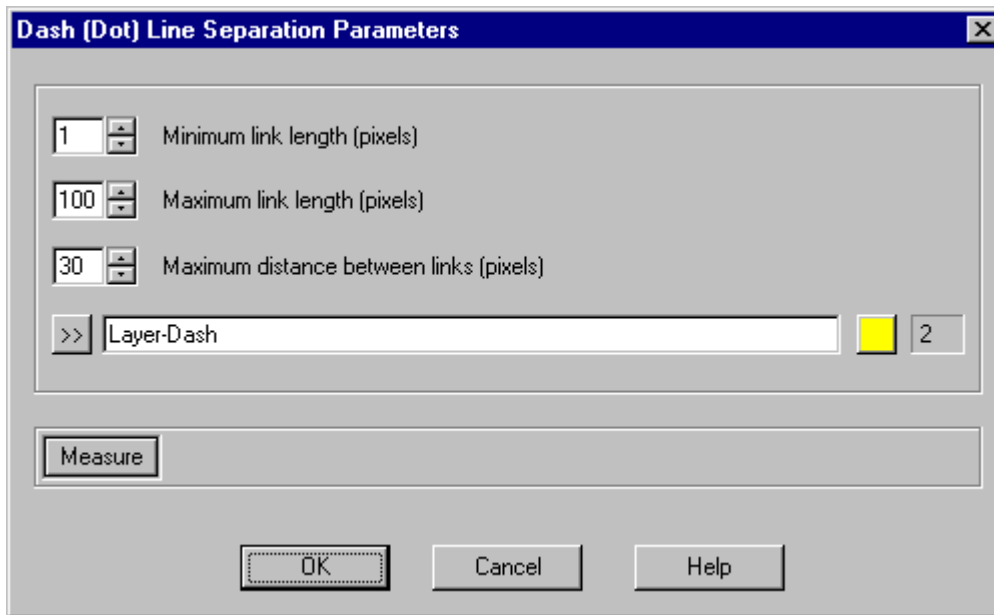
In this dialog box you may set up parameters for automatic separation of dash, dot and dash-dot lines from other recognized objects. Note that EasyDone does not substitute such lines by corresponding AutoCAD primitives, like in other cases, because this process requires participation of user (at least to select for each type of dash lines on raster the most appropriate Linetype from AutoCAD library).
The main goal of dash separation is to assign dash, dot and dash-dot lines to a separate layer, so user could freeze this layer and avoid removing of recognized dash lines manually dash by dash.

Minimum Link Length
This parameter defines minimum length of a single line, which should be interpreted as a link inside dash-line. All single lines, which length is less than this value, will be interpreted as not belonging to dash/dot lines.
Note: If you need to recognize dot or dash-dot lines, this parameter should be set to value "1", otherwise dots will be removed as background noise.

Maximum Link Length
This parameter defines maximum length of a single line, which should be interpreted as dash/dot line link.

Maximum Distance Between Links
This parameter defines maximum allowed distance between ends of two adjacent links.
If such distance exceeds the value of this parameter, it will be interpreted as end of dash/dot line.

Note: Some actual links may be not interpreted as belonging to dash/dot lines in the case when EasyDone has doubts on this matter.

Hatch Separation Parameters
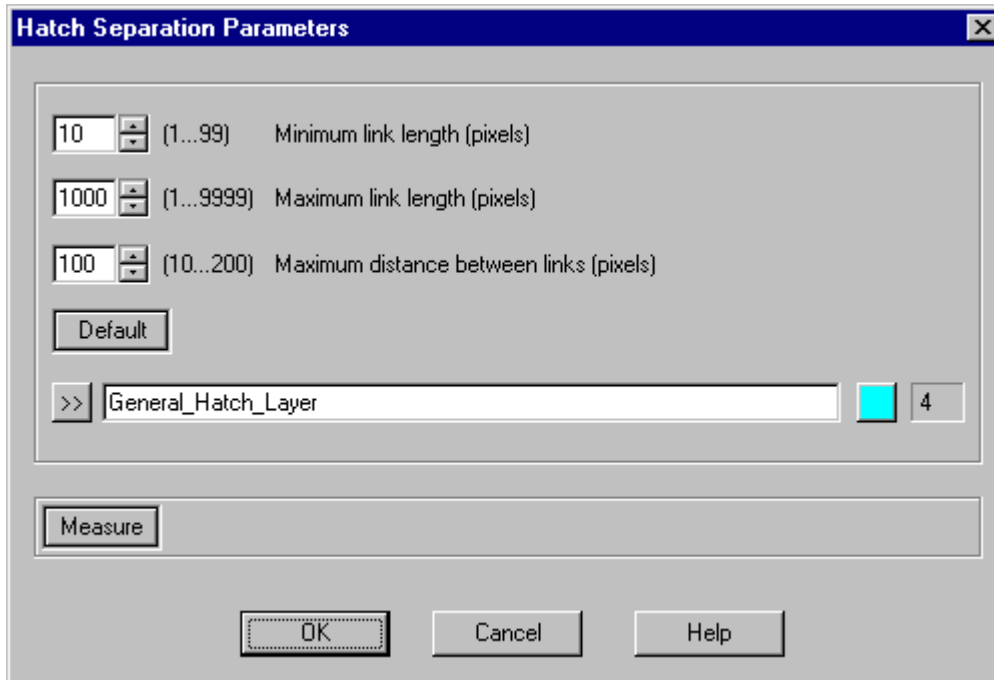
In this dialog box you may set up parameters for automatic separation of hatch lines from other recognized objects. Note that EasyDone does not substitute such lines by corresponding AutoCAD primitives, like in other cases, because this process requires participation of user (at least to select for each type of hatch lines on raster the most appropriate type of hatching from AutoCAD library).
The main goal of hatch separation is to assign hatch lines to a separate layer, so user could freeze this layer and avoid removing of recognized hatch lines manually link by link.

Minimum Link Length
This parameter defines minimum length of a single line, which should be interpreted as a link inside hatch object. All single lines, which length is less than this value, will be interpreted as not belonging to hatch object.

Maximum Link Length
This parameter defines maximum length of a single line, which should be interpreted as a hatch link.

Maximum Distance Between Links
This parameter defines maximum allowed distance between two adjacent hatch links.
If such distance exceeds the value of this parameter, it will be interpreted as an edge of hatch object.

Default
This button sets up the most common values for hatch separation parameters. You may use these values as a basis for modification of hatch separation parameters according to the specific of some particular image.

Note: Some actual links may be not interpreted as belonging to hatch object in the case when EasyDone has doubts on this matter.

Arrowhead Recognition Parameters

Note: This mode is designed for recognition of triangle solid arrowheads only. "Linear" arrowheads will be recognized as regular graphics (see illustration).
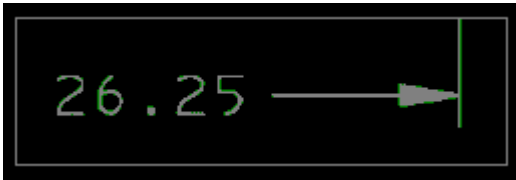
Recognition of triangle solid arrowhead
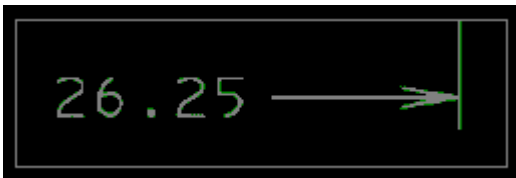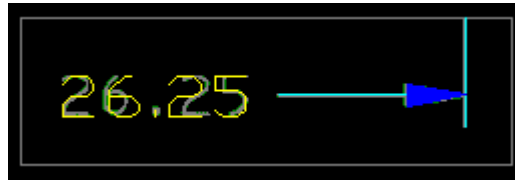
Recognition of "linear" arrowhead

Trim Underlying Vector

This check box turns ON or OFF the automatic trimming of the underlying vector in the area covered by arrowhead.

Note: If arrowhead is too narrow (a few raster pixels wide), trimming is not possible.

If this check box is ON, arrowheads will be presented in the output data as an outline and not filled.

If this check box is ON, the table of arrowhead sizes will be disabled and the output size of each arrowhead will be determined automatically. In this case program automatically classifies all recognized arrowheads into different size classes and provides exactly the same size of arrowheads inside each class.

This parameter limits maximum arrowhead length when "Automatic Size" is turned ON. It is recommended do not set this parameter to unnecessary large value, otherwise program may try recognize arrowheads on solids.

Table of arrowhead sizes
This table is enabled if "Automatic Size" parameter is turned OFF, and allows defining of classes for different arrowhead sizes. Each class sets up output pixel length of arrowheads, which length on raster is between minimum and maximum value for this class, and a width/length ratio of output arrowheads.

TIP: This table may be used in a partial manner by entering "0" in Output Length field for the class to be defined with automatic size option.

Outlining Parameters
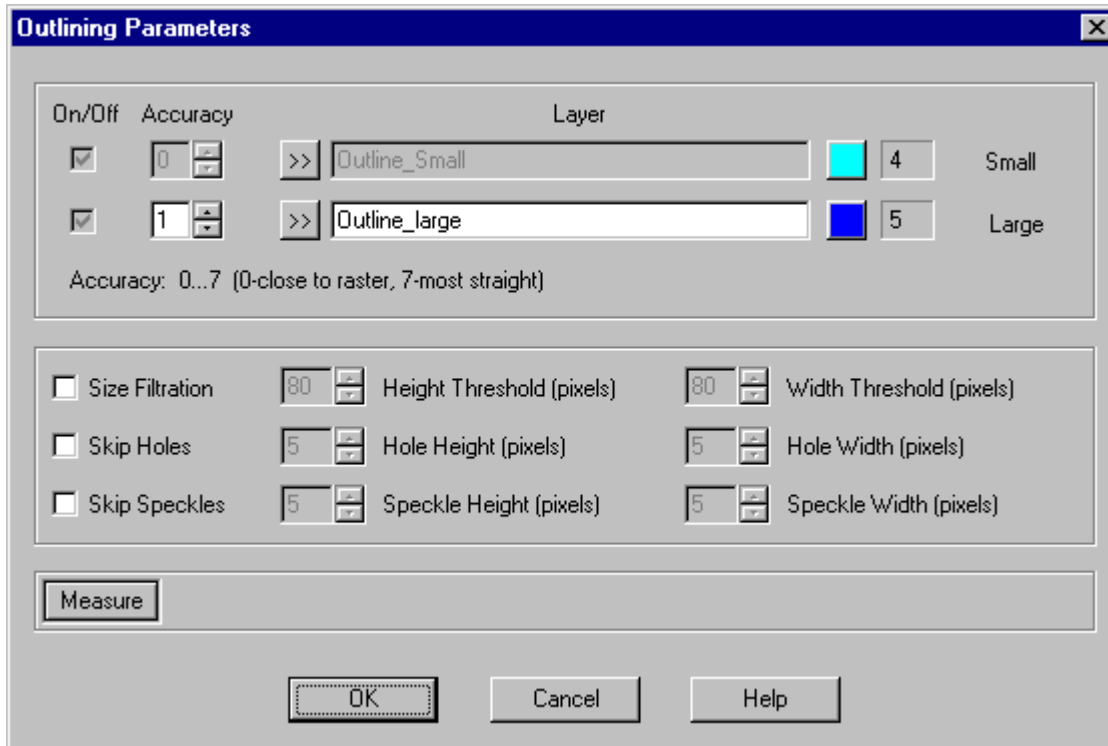
In this dialog box you may set up parameters for raster outlining. Outlining is an alternative mode to text and graphics recognition, in which output data will contain only polylines of outlined contours of raster objects independently on whether it is text or graphics. Note that you may use outlining in combination with text and graphics recognition, if start it not on the whole image but only on that image part, which should be outlined.

Size Filtration
This option allows to separate outlined raster objects to "small" and "large" classes and to set up different parameters (if necessary) for each class.
If this option is turned OFF, all outlined objects will belong to large class.
If this option is turned ON, the threshold of height and width between small and large classes is defined by parameters "Height Threshold" and "Width Threshold". When both height and width of raster object do not exceed corresponding thresholds, it is assigned to small class, otherwise to large class.

Skip Holes
If outlined objects have some meaningless holes inside, it is possible to skip such holes.
The height and width of holes, which should be skipped, is defined in parameters "Hole Height" and "Hole Width".

Skip Speckles
If there are some meaningless speckles on the raster image, it is possible to skip outlining of such objects.
The height and width of speckles, which should be skipped, is defined in parameters "Speckle Height" and "Speckle Width".

Text Recognition Parameters

Directions

This parameter defines in which directions EasyDone should try to recognize text. You may set up any combination of these directions, but it should be in correspondence with actual directions of text on your document.

For instance, if document has only horizontal and 90 degrees vertical text but you set up value "Any", it may unnecessarily decrease quality of recognition, especially for standalone characters. Many of standalone (and not only) characters may be interpreted different ways depending on their actual direction (for example digit "6" in horizontal direction means the same as digit "9" in 180 degrees direction, etc.).

On the other hand, if document has text written, for example, with 45 degrees angle but value "Any" is not checked, program will try to recognize text only in orthogonal direction and result will be wrong.

Output Form

This parameter allows recognition of not only single words but text strings as well. Value "Sentence" means particular case of string recognition - string ending with dot. Please pay attention that if you select "String" or "Sentence", then only the left point of the first word and the right point of the last word will exactly fit in raster, unlike selection "Word". If you need to get recognized text as strings and at the same time each word must coincide with raster origin, it is recommended to recognize text with selection "Word" and then create strings manually, using command "String" in Recognition Result Editor. It is also not recommended to use selections "String" and "Sentence" for cases when lines of text are written too close to each other (especially if they are not horizontal) because EasyDone may accidentally jump from one line to another and produce incorrect result.

This parameter defines whether EasyDone should recognize both upper- and lowercase text. Recognition of capital characters is enabled by default. Many drawings have capital letters only and it may be useful to turn this parameter OFF to get more reliable recognition result.

This parameter enables using of right insertion point of recognized word (string). This point coincides with the right bound of corresponding word (string) on raster. See illustration below for different result of recognition with this parameter turned ON and OFF.

Result with parameter turned OFF

Result with parameter turned ON

It is recommended to keep this parameter turned ON, unless you want to specify different Font Width for different classes. Note that it is not possible to change width of text entities, recognized without right insertion point, with frame-based operation in Recognition Result Editor.

Text Classes
This group of parameters allows defining different recognition mode, output font height, width factor and oblique angle of output font, and layer/color for text entities with different raster height.

This parameter specifies recognition mode for each class. Value "1" defines automatic recognition, "2" semi-automatic recognition, and "0" will leave text of the corresponding class without text recognition. If you start text+graphics recognition, text from classes with "0" mode will be recognized as graphics, otherwise it will be skipped.
TIP: It may be helpful in some cases to use combination of modes. For instance, you may set up semi-automatic recognition for small text (because there is higher probability of recognition errors) and automatic recognition for other classes.

These parameters define for each class minimum and maximum height limits in pixels of text on raster. You should be careful when defining large value for the maximum limit because as bigger is this parameter as higher is probability that some small graphical object, which size is lower than this limit, will be interpreted as text and will be recognized incorrectly.

This parameter allows obtaining of result of text recognition with exact size of output fonts. If you know what size output font should have in World Coordinates (WC), you may define it for each class using this parameter. If you do not need this feature, you may set up the value of parameter to zero and the output height of each recognized text entity will be defined automatically according to the height of the raster origin.

TIP: If you set this parameter with some non-zero value, then all words of the corresponding class will have exactly the same output height, otherwise each word will have its own output height according to the height on raster.

This parameter specifies the oblique angle of output font and may have as positive as negative value. If value is negative, then font will be inclined to the left side. If value is positive, then font will be inclined to the right side. See illustration below.

Font Angle with value –20 degrees

Font Angle with value –45 degrees

Font Angle with value +20 degrees

Font Angle with value +45 degrees

Font Width

This parameter is enabled only if parameter "Automatic Width" is turned OFF. It specifies width factor of output font. Entering a value less than 1.0 condenses the text. Entering a value greater than 1.0 expands it.

Text Style

This parameter defines name of text style, which will be assigned to recognized text entities of the corresponding class. If drawing does not have this style, it will be created. You may use different text styles for different classes to assign desired font to recognized text in the corresponding class (see the topic "Recognition Process/Assigning Different Fonts").

Control Options

This button opens "Control Options" dialog box, in which you may fine-tune some additional parameters of text recognition, if necessary.

Interpretation

This button opens "Interpretation Options" dialog box, in which you may fine-tune some additional parameters of text recognition, if necessary.

International Character Set

This button opens "International Character Set" dialog box, in which you may turn on recognition of non-English characters (see the topic "Recognition Process/Recognition of non-English Characters"). Note that by default (after installation) recognition of these characters is turned off.

Measure button

This command is used to measure height, width, and associated angle of different raster objects.
The measured values in pixels and World Coordinates (WC) will appear on the bottom line of the current dialog box. These values may be required to set up some graphics and text parameters.
See the topic "Additional Operations/Menu "Measure Size" for information how to use command.

Text Parameters: Control Options
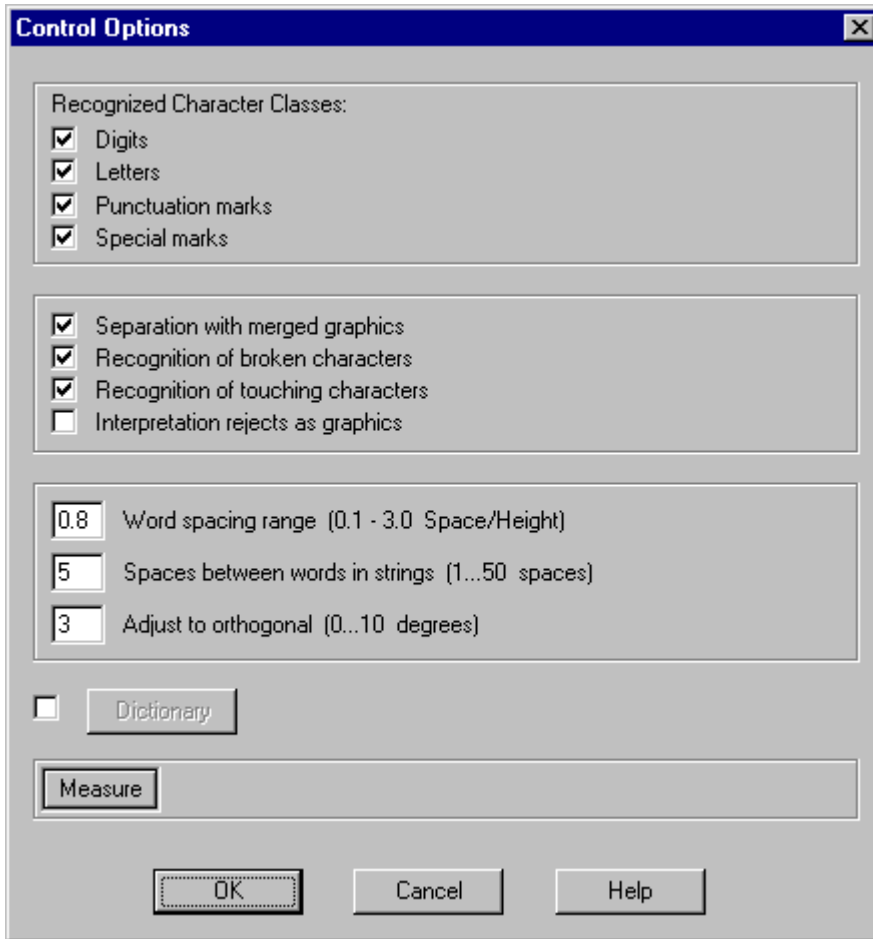
Recognized Character Classes
The parameters of this option can be set up to recognize:

"Digits"   1234567890
"Letters"   ABCDEFGHIJKLIMNOPQRSTUVWXYZ
"Punctuation Marks"   ,.'";:?!
"Special Marks"   #%&*()-_+=[]| /<>

When some parameter is turned OFF, program will not consider characters of corresponding class as object for text recognition. For example, if document contains only numbers, parameter "Digits" may be turned ON and others turned OFF for faster and more reliable recognition.

Separation With Merged Graphics
This parameter enables recognition of characters, which touch graphics lines. If it is turned OFF, then such characters will be recognized as graphical objects. See illustration below.

Parameter is turned ON

Parameter is turned OFF

Recognition of Broken Characters
This option allows recognizing of broken characters with up to 3 gaps in each.

Recognition of Touching Characters
This option allows recognizing of up to 3 characters touching each other.

Interpretation Rejects as Graphics
This parameter determines what the recognition program should do with rejects (reject means situation when program cannot recognize some character reliable).
When this parameter is turned OFF, reject will be marked by character "~" (tilde), that helps to find it in Recognition Result Editor and type instead of this mark the required character.
When ON, the corresponding raster area will be interpreted as background noise or (in the case of text+graphics recognition) as graphics.

Word Spacing Range
This parameter represents the maximum space between characters (as ratio of their height), for which recognition program will join these characters into one word. The parameter value can be set up from "0.1" to "3.0", but for most images "0.8" value is recommended.
If too many words are incorrectly split up by program into a few parts, you may increase the parameter value. If words are wrongly joined, you may decrease the value. However, it is not recommended to change value "0.8" without actual necessity because this value passed through many tests and always provided the most reliable result.

Spaces Between Words In Strings
This parameter defines maximum number of spaces between two adjacent words in text strings, if "String" or "Sentence" output form is selected in Text Recognition Parameters dialog box.
If the number of spaces between such words exceeds parameter value, the first string will be ended and the second string will be started from the next word.

Adjust To Orthogonal
This parameter enables adjusting of recognized text entities to horizontal or vertical direction. The parameter value can be set from 0 to 10 degrees. If the recognized text deviates from the horizontal or vertical axis by less than this value, this text will be rotated to the horizontal or vertical axis.

Dictionary check box
This toggle enables or disables the use of a dictionary.

Dictionary button
This button invokes the "Text Recognition Dictionary" dialog box with settings for such advanced function as auto-correction of errors and rejects using a dictionary prepared by the user.

Measure button
This command is used to measure height, width, and associated angle of different raster objects.
The measured values in pixels and World Coordinates (WC) will appear on the bottom line of the current dialog box. These values may be required to set up some graphics and text parameters.
See the topic "Additional Operations/Menu "Measure Size" for information how to use command.

Text Parameters: Interpretation Options

Interpretation Options

Recognition of Single Characters

`4` Isolated from graphics

`4` Merged with graphics

(1-any angle, 2-horizontal, 3-reject, 4-graphics)

Recognition of Single Vertical Line '|'

`0` Isolated from graphics

`0` Merged with graphics

(0-graphics, 1-as '1', 2-as '|')

Contextual Recognition of Twin Characters

`1`   0 vs. O

`1`   1 vs. I

`1`   2 vs. Z

`1`   5 vs. S

`1`   6 vs. G

`1`   8 vs. B

(0-context off, 1-context on, 2-digit, 3-letter)

[OK]   [Cancel]   [Help]

Recognition of Single Characters
This parameter defines what program should do with a single character in two cases: when it is isolated from graphics and when it is merged with graphics.
One of these four options may be selected:
"1" - recognize with direction "Any" (see the topic "Text Recognition Parameters");
"2" - recognize in horizontal direction (from left to right);
"3" - mark as reject;
"4" - interpret as background noise or recognize as graphics (in the case of text+graphics recognition).

Recognition of Single Vertical Line "|"
This parameter specifies what the program should do with a single vertical line "|" when it is isolated from graphics and when it is merged with graphics.
One of these three options may be selected:
"0" - interpret as background noise or recognize as graphics (in the case of text+graphics recognition);
"1" - recognize as "1";
"2" - recognize as "I".

Contextual Recognition of Twin Characters
This parameter separately sets the mode of contextual interpretation for the following "twin" pairs:
       "0" vs. "O"
       "1" vs. "I"
       "2" vs. "Z"
       "5" vs. "S"
       "6" vs. "G"
       "8" vs. "B"

You may select:
"0" - context is OFF, contextual recognition of twin characters is not used;
"1" - context is ON,   contextual recognition of twin characters is used based;
"2" - a letter will be always substituted by a corresponding digit ("S" by "5", "G" by "6" and so on);
"3" - a digit will be always substituted by a corresponding letter ("5" by "S", "6" by "G" and so on).

Recommended parameter value is "1".

Text Recognition Dictionary

This option allows the program to correct the result of text recognition using a special "dictionary", that is a text file prepared by user with any word-processing program, for instance Notepad.
Using of dictionary may increase reliability of text recognition on documents, which have some pre-defined text constructions known to user, such as for example technical specifications, GIS bearings and so on.

The Dictionary file contains words or patterns, which will be compared with words in the result of text recognition. When words or patterns in the Dictionary file match the result of text recognition, the patterns are used for automatic correction of errors/rejects. The Dictionary file may consists of up to 512 lines, every one of which corresponds to one word or pattern.

Each line has format X <Y> beginning from the first position.

Values for "X" may include:
"1" - for "loose" matching;
"2" - for "medium" matching;
"3" - for more accurate matching (with minimum mismatches);
"0" - to ignore the line.
The "X" value is followed by a space and the pattern for "Y" value in the brackets: "<" and >".

The "Y" pattern consists of characters without any space(s) between them.
Note: Do not use lowercase letters in the "Y" pattern.
For a word or pattern to match a word from the text recognition result it is necessary that every character in the pattern correspond to the character in the same position in the matching word.

The following special characters may be used in the pattern:
"^" represents "any digit";
"|" represents "any letter";
"~" represents "any character".

An example of dictionary file

The following Dictionary file was used for more reliable text recognition of images where the same words were often repeated.
1 <POWER>
1 <DEVICE>
1 <FIELD>
1 <CODE>
1 <FROM>
1 <PHYS>
1 <EQUIPMENT>
1 <DWG:>
1 <HTR>
1 <OR>
1 <ISSUE****>
1 <****REVISED>
1 <ROUTE:>
1 <DIRECT>
1 <BURIAL>
1 <X1073C>
1 <NO/C>
1 <PIF-^-^^>
1 <PIF-^-^>
1 <PIF-^>
1 <E19-36>
1 <MINI>
1 <CTL>
1 <CG>
1 <STA>
1 <E25>
1 <RECEPTACLE>

1 <CIRCUIT>
1 <NUMBER>
1 <CABLE>
1 <WIRE>
1 <SIZE>
1 <SPEC>
1 <SYSTEM>

Dictionary file also allows user to customize some semantic context.
The following example will work for images that do not have words containing two digits with the letter "B" at the end.

1 <^^8>

The line above, added to a Dictionary file, will automatically change an incorrectly recognized "B" to the correct "8" in these words.

Recognition Result Editor Parameters
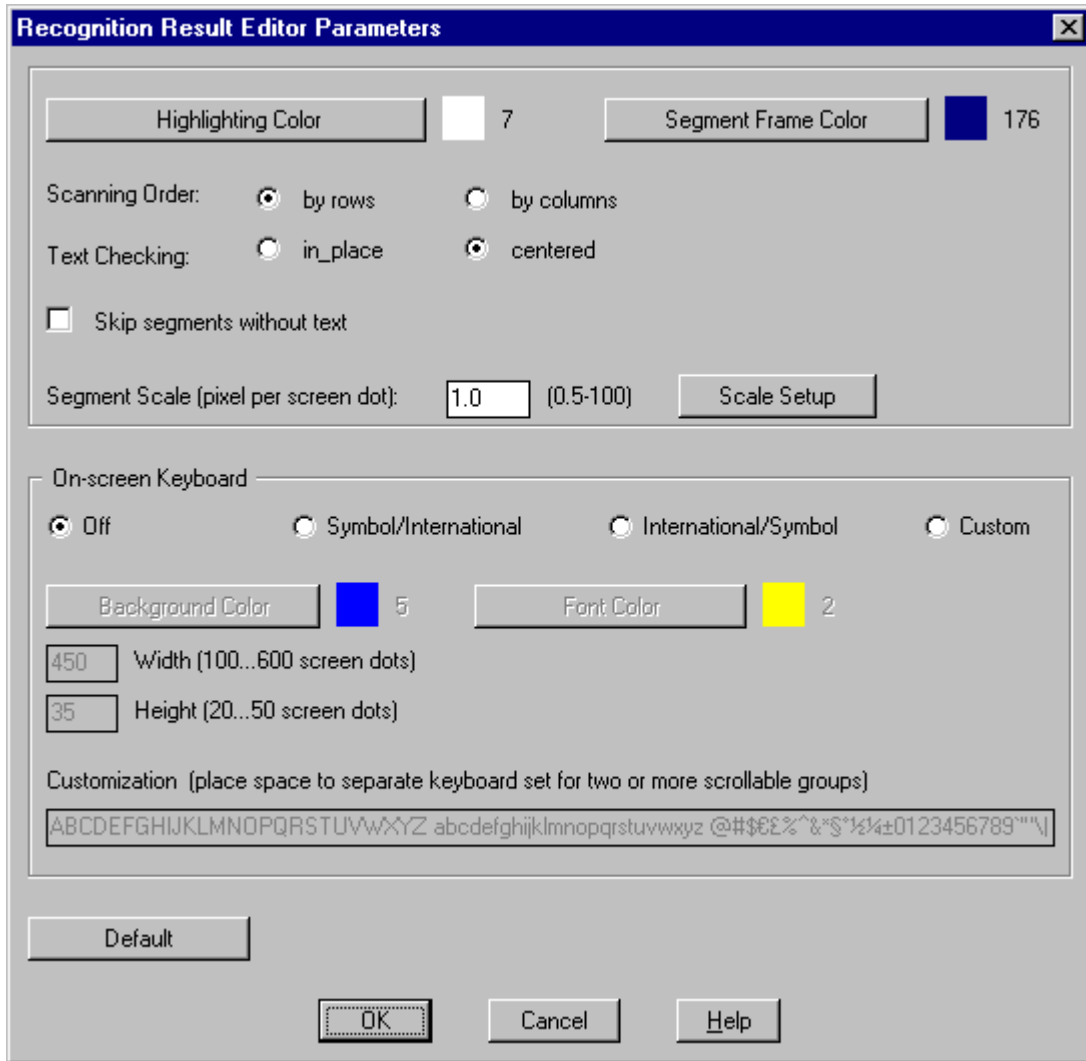
Highlighting Color
This parameter defines the color used to highlight the "marked" text entities, which are already checked and/or edited. White color is recommended if raster has gray color.

Segment Frame Color
This parameter defines the color used in SEGMENT mode for segment frame. It is recommended to select this color so, that it cannot be mixed up with recognized graphics lines (the recommended value is "176"). This color is also used for polyline highlighting in "Polyline Follow" operation when you temporarily switch from the followed polyline to some neighbor object to edit it.

Scanning Order
Choose between "by rows" and "by columns" to select the order, in which the consequent segments will be shown in SEGMENT mode.

Text Checking
This parameter defines how text entities will be shown in AutoText mode for checking and editing.
"In_place" - a new view window is used to put more than one text entity on screen and then highlight them one by one without changing of the current view.
"Centered" - a new view window is used to put every text entity in the center of the screen (recommended).

Skip Segments Without Text
When you work in SEGMENT mode, this option allows skipping of segments, which contain only recognized graphics and do not contain recognized text.

Note: The first segment shown in SEGMENT mode is always the top left segment, even if it does not contain recognized text, graphics or raster at all (it is necessary to initiate SEGMENT mode).

Segment Scale
This parameter is counted as number of raster pixels per screen dot, and defines the size of the segment when you work in SEGMENT mode.
It is recommended to avoid unnecessary magnification - it may dramatically increase the number of segments to be shown. Usually the best value is "1.0" (one raster pixel per screen dot).
TIP: If you want to place the whole drawing onto only one segment, you may use any large value for this parameter (the maximum allowed value is "100.0").

Scale Setup
You may use this option to set up the Segment Scale parameter automatically, according to the scale of the current view. The current view (part of the drawing, which is currently visible) will be considered as one segment, so the segment scale will be counted automatically according to it.
You may zoom the current view to the desired scale, then call Recognition Result Editor Parameters dialog box and use "Scale Setup" option.

When you press "Scale Setup" button, the dialog box temporarily disappears and the following messages appear in the command line:

Current Scale (raster pixels per screen dot) : xxx
[SCALE]Value/Quit/<Save_scale>

If you want to save the current scale, just press Enter or right mouse button. If you want to adjust it, type "**v**" for selection "Value" and then type value of the new scale. The current view will be changed according to this value and you may then save it or change again.

On-screen keyboard
The "On-screen keyboard" is an auxiliary tool, which allows you to select some character by mouse from the specially organized list instead of typing it from keyboard (see illustration below).

The important feature of the on-screen keyboard is that it allows direct input of characters and symbols, which are not presented on standard keyboards. On-screen keyboard may be also helpful if recognized text does not contain many errors or rejects, so you do not have to switch from mouse to keyboard while you check the recognized document.

To invoke the on-screen keyboard for the highlighted word, you may click some character (if you are going to edit it) or space between characters (if you are going to insert a new character) and then move mouse cursor across the top or bottom side of the highlighting frame. On-screen keyboard will appear above (under) the frame and in the same direction. To hide the on-screen keyboard, move cursor outside of it.

TIP: If you clicked a character or selected a group of characters inside the highlighted word, it is recommended to click it one more time to fix your selection (in this case the inner selection frame becomes dashed, like on the illustration above). If you do not fix it and try to invoke the on-screen keyboard, the mouse movement may unintentionally change selection.

The set of characters, available with on-screen keyboard, may be divided into several groups to reduce the keyboard length. To scroll from current group to the next one, you need to press right mouse button. This scrolling is cyclic, so after the last group the first group appears again.

EasyDone is supplied with two pre-defined on-screen keyboard sets, which you may enable with the corresponding radio button. You may also define your own character set for the on-screen keyboard. If you click radio button "Custom", then edit-field "Customization" is enabled and you may type in this field your own character set. To divide your set into groups, separate each group by **one** space.

Selective Mode

Selective Mode is one of two main modes, in which Recognition Result Editor works (another one is Segment Mode). In Selective Mode you may click any recognized object (text entity, polyline, solid, or circle) and then edit it with commands of Recognition Result Editor. You may also start certain operations in Selective Mode (the difference between terms "command" and "operation" is that "command" usually implements a single editing action, whereas "operation" is most likely a kind of self-dependent editing sub-process).

To start Recognition Result Editor in Selective Mode you may use menu "EasyDone/Recognition Result Editor/Selective Mode" or corresponding EasyDone toolbar button (see below).

After that the following message appears in the command line:

[SELECTIVE]Amend_corners/String/Obj/Zoom/couNt/Help/Esc

Amend_corners is special graphics editing operation for automatic or interactive improvement of recognized lines, which have distortions in corners between them or require orthogonal adjustment. To invoke this operation, type "**a**" from keyboard.

String is special text editing operation, which allows joining of single recognized words (or another strings) into one string that will be presented in output data as one text entity. To invoke this operation, type "**s**" from keyboard.

Obj is editing operation, which allows inserting of new AutoCAD entities, such as text, polyline, solid, or circle. The reason to use this operation instead of similar AutoCAD capabilities is that inserted objects are assigned to corresponding classes, defined in recognition parameters. To invoke this operation, type "**o**" from keyboard.

Zoom is just the replica of the AutoCAD command "Realtime Zoom". It allows zooming or panning of the drawing without leaving Selective Mode. To invoke it, type "**z**" from keyboard. To quit **Zoom** command, press right mouse button and select item **Exit** in the pull-down menu.

couNt command displays the statistic information regarding the number of recognized objects on the whole drawing and which part of them is already checked or edited. To invoke this command, type "**n**" from keyboard.

To quit Selective Mode, press Esc or type "**e**" from keyboard.

If you want to discard highlighting of objects, which are marked as already checked/edited, you should remove previous editing marks, otherwise these objects will become highlighted every time when you start Selective or Segment Mode. You may use for that menu "EasyDone/Recognition Result Editor/Remove Marks" or corresponding EasyDone toolbar button (see below).

After that the "Remove Editing Marks" dialog box will appear. You may specify in this dialog box removing of resetting of edit marks and/or deleting of contours of solids.

Resetting of editing marks discards all highlighting of edited objects and allows starting of Segment Mode from the beginning. Removing of editing marks deletes from the drawing all extended data related to Recognition Result Editor. Note that if you apply this option, you will not be able to use Recognition Result Editor with this drawing any more.

Deleting of contours of solids remains solids only in the form of SOLID hatch pattern (each solid is recognized by EasyDone as hatch pattern plus polyline(s) of the contour and holes, if they exist). Note that if you apply this option, you will not be able to use Recognition Result Editor for editing of solids on this drawing any more.

Segment Mode

Segment Mode is one of two main modes, in which Recognition Result Editor works (another one is Selective Mode). In Segment Mode program automatically divides current drawing into a sequence of adjacent segments, which will be shown one by one for checking of recognition result and editing, if necessary. The reason for dividing of a drawing into segments is that scale of the whole drawing usually does not allow seeing of all details of recognized objects, so it is not possible to check or edit them without additional magnification. Contrary to that, in Segment Mode you may select any convenient size of segments, so you may check or edit all objects on the segment without changing of current view (see the topic "Recognition Result Editor Parameters" for information how to set up segment scale).
The second advantage of Segment Mode is that all segments are shown sequentially, so you may be sure that nothing is missed without checking and, at the same time, you did not check the same place twice.
The third advantage is that Segment Mode has "AutoText" sub-mode, in which all text entities, located on the segment, will be shown one by one for their checking and editing.
In addition to the "AutoText" sub-mode you may click any recognized object and then edit it with commands of Recognition Result Editor, like in Selective Mode.

To start Recognition Result Editor in Segment Mode you may use menu "EasyDone/Recognition Result Editor/Segment Mode" or corresponding EasyDone toolbar button (see below).

After that the following message appears in the command line:

[Define Editing Zone]Whole_image/New/<Whole_image>

If you want to start Segment Mode on some part of the drawing (selection **New**), type "**n**" and press Enter, otherwise press Enter or right mouse button for default selection **Whole_image**.

After you complete your selection, the following message appears in the command line:

[SEGMENT]auto_Text/String/Obj/Zoom/Change_scale/vieW_done/View_seg/Back/couNt/Help/Esc

auto_Text command allows to start "AutoText" sub-mode on the current segment, if you closed it before all text entities on this segment were shown (in other cases you do not need specially start "AutoText" sub-mode because it starts automatically when you move to the next segment and press Enter or right mouse button). To invoke this command, type "**t**" from keyboard.

String is special text editing operation, which allows joining of single recognized words (or another strings) into one string that will be presented in output data as one text entity. To invoke this operation, type "**s**" from keyboard.

Obj is editing operation, which allows inserting of new AutoCAD entities, such as text, polyline, solid, or circle. The reason to use this operation instead of similar AutoCAD capabilities is that inserted objects are assigned to corresponding classes, defined in recognition parameters. To invoke this operation, type "**o**" from keyboard.

Zoom is just the replica of the AutoCAD command "Realtime Zoom". It allows zooming or panning of the drawing without leaving Segment Mode. To invoke it, type "**z**" from keyboard. To quit **Zoom** command, press right mouse button and select item **Exit** in the pull-down menu.
Note: Unlike the same command in Selective Mode, you may use command Zoom only for browsing of the drawing because, after you quit command Zoom, Segment Mode automatically returns to the current segment. If you want to move to some place outside of the current segment and edit there something, you may use scroll-bars and command "Change_scale" (see below).

Change_scale is special viewing command, which allows changing of scale of the current view dynamically. Note that this "scale" does not have any relation to the Segment Scale and does not affect it. To invoke this command, type "**c**" from keyboard.

vieW_done is special viewing command, which shows the whole drawing with highlighting of the segments, which were already passed, and entities, which were already checked or edited. You may see

also location of the current segment on the drawing. To invoke this command, type "**w**" from keyboard. To return to the current segment, press Enter or right mouse button.

View_seg command returns the view to the current segment. It may be used if you moved outside the current segment by command Change_scale or scroll-bars. To invoke this command, type "**v**" from keyboard.

Back is command, which returns Segment Mode to the previous segment. To invoke this command, type "**b**" from keyboard.

couNt command displays the statistic information regarding the number of recognized objects on the whole drawing and which part of them is already checked or edited. It includes also the total number of segments and the number of segments, which were already passed. To invoke this command, type "**n**" from keyboard.

Segment Mode always starts from the top left segment, even if it does not contain recognized text, graphics or raster at all (it is necessary to initiate Segment Mode).
When program enters a new segment, you have two choices: to start "AutoText" sub-mode or to click some object and edit it.
To start "AutoText" sub-mode, you need to press Enter or right mouse button. When "AutoText" sub-mode is complete (that is when all text entities on the current segment are shown), program shows the whole segment again. You may then click some object and edit it or move to the next segment. To move to the next segment, you need to press Enter or right mouse button. When program moves to the next segment, the previous segment is highlighted with special "grid". All objects, located entirely in the already passed segment(s), are highlighted with special color defined in Recognition Result Editor Parameters dialog box. This highlighting helps to distinguish checked or edited objects and remains until you quit Recognition Result Editor.

When you have passed through all segments, the Segment Mode is closed automatically.
If you want to abort Segment Mode before that, press Esc or type "**e**" from keyboard.

If you want to start Segment Mode from the beginning on the drawing, which was already edited in Segment or Selective Mode, you need first to remove previous editing marks. You may use for that menu "EasyDone/Recognition Result Editor/Remove Marks" or corresponding EasyDone toolbar button (see below).

After that the "Remove Editing Marks" dialog box will appear. You may specify in this dialog box removing of resetting of edit marks and/or deleting of contours of solids.

Resetting of editing marks discards all highlighting of edited objects and allows starting of Segment Mode from the beginning.

Removing of editing marks deletes from the drawing all extended data related to Recognition Result Editor. Note that if you apply this option, you will not be able to use Recognition Result Editor with this drawing any more.

Deleting of contours of solids remains solids only in the form of SOLID hatch pattern (each solid is recognized by EasyDone as hatch pattern plus polyline(s) of the contour and holes, if they exist). Note that if you apply this option, you will not be able to use Recognition Result Editor for editing of solids on this drawing any more.

Editing of Text

To make some text entity available for editing, you need to click it by left mouse button. After that the selected text entity becomes marked with special highlighting frame (see illustration below).

If you click text entity in Selective Mode, the following message appears in the command line:

[TEXT]%Undo=Tab %Class=F3 %Join=F4 %Split=F5 %Del=space %Rid=F6 %Obj=F7 <-->F8 %Magn=F9 %Zoom %Help=F11 /<mark>

When you work in Segment/AutoText mode, the following message appears in the command line:

[TEXT]%Undo=Tab %Class=F3 %Join=F4 %Split=F5 %Del=space %Rid=F6 %Obj=F7 <-->F8 %Magn=F9 %vieW_done=F10 %View_seg %bacK %couNt %Help=F11 /<mark>

TIP: In "AutoText" sub-mode you have not click text entities, however you can do it if you want to edit not the text entity, which is currently shown by "AutoText" sub-mode, but some another one.

Note that character **%** is used as a control character to distinguish command from typed text. If you need to type character **%** in your text, it is necessary to type it twice (**%%**).

You may edit text entity three different ways:
1 With direct editing of characters;
2 With frame-based operations;
3 With commands listed in the command line.

Direct editing of characters
If you want to replace some reject (reject is marked with character "~" tilde) or wrongly recognized character, you need to click it by left mouse button. After that the inner selection frame appears around the selected character.

You may type one or more characters, which will substitute the selected one.

If you typed more than one character, the width of all characters will be automatically adjusted (if text entity was recognized with right insertion point).

You may also select not one but several characters at once. To do so, click the leftmost or rightmost character and then move mouse cursor to capture other characters, which you want to select.

After that it is recommended to fix this selection by pressing left mouse button one more time, because otherwise the selection frame may be unintentionally changed by movement of mouse cursor. After you press left mouse button, the selection frame becomes dashed.

Now you may type required characters, and mouse movement will not affect this process. We will type lowercase characters for the following sample.

If you want not to replace but insert new character(s), you may click by left mouse button the desired position between existing characters, and the dashed vertical insertion marker will appear there. We will insert this marker between characters "r" and "i" for the following sample.

Then you may type one or more characters – they will be inserted from the left side of the insertion marker. We will insert digit "2" for the following sample.

TIP: You may use key "Backspace" on keyboard to delete character from the left side of the insertion marker, like in any standard editor. Unfortunately AutoCAD does not provide program access to the "Delete" key, so it is not possible to use it as well. To delete some character or space you have to use Backspace key, or you may select deleting character along with the left or right adjacent neighbor and then to type this neighbor character alone. For illustration, we will select character "r" and digit "2" with the goal to delete this digit.

Then we will type character "r" and it will result in deleting of digit "2".

You may also replace all characters in the highlighted text entity at once. To do so, do not click anything in the highlighted text entity (otherwise the insertion marker or selection frame will appear) but simply type all required characters from keyboard. If you press the space bar instead of typing of new characters, it will take the same effect as command **%Delete**.

TIP: In some cases it may be more convenient not to type characters from keyboard, when you edit text entity, but to use so-called "On-screen keyboard". See the topic "Recognition Result Editor Parameters" for details.

Frame-based operations
The highlighting frame not only highlights the text entity, which is currently available for editing, but allows also to apply five frame-based editing operations. Each operation may be invoked by clicking of the corresponding part of the frame.

TIP: See for illustration of frame-based operations "Demo Tutorial. Step 7".

To **Move** text entity, click by left mouse button the left vertical side of the frame, move text entity to the new position (or type new coordinates) and press left mouse button again to complete operation.

To **Rotate** text entity, click by left mouse button the bottom side of the frame, rotate text entity to the desired angle (or type new angle value) and press left mouse button again to complete operation.

To change **Height**, click by left mouse button the top side of the frame, move cursor to the desired new height (or type new height value) and press left mouse button again to complete operation.

To change **Width**, click by left mouse button the right vertical side of the frame, move cursor to the desired new width and press left mouse button again to complete operation.
Note: If text entity was recognized with option "Automatic Width" turned OFF (see the topic "Text Recognition Parameters"), you cannot change width of such text entity with frame-based operation, and the right vertical side will be absent in the highlighting frame. However you may change option "Automatic Width" individually for this text entity by command **%Class** (see the topic "Text Entity Editing Commands/Command: %Class"), and then to change the width by frame-based operation.

If text entity was recognized by mistake upside-down, you may flip it with frame-based operation.
To **Flip** text entity, click any point on the triangle lug of the highlighting frame.
TIP: This lug shows the direction, in which text entity was recognized, and must be located at the right end of it. When you type text after semi-automatic text recognition, pay attention where the lug is located, and flip text entity, if necessary, before you begin typing characters for this entity.

Command line commands
Each command may be invoked by the corresponding functional key (if assigned) or by typing of character "**%**" and the character, which is capitalized in the name of the command (for instance character "w" in the command "vieW_done"). Character "**%**" is used by program to distinguish character, which invokes some command, from characters, which you use for typing.

Command **%Undo** or key **Tab** undoes result of the last editing operation.

Command **%Class** or key **F3** allows changing of the following parameters, assigned to the text entity at the time of recognition process: layer/color, font height, font oblique angle, font width, text style, and the setting "Automatic Width", which defines whether text entity has the right insertion point. See the topic "Text Entity Editing Commands/Command: %Class" for details.

Command **%Join** or key **F4** allows joining of the highlighted entity with another text entities, from which it was separated by mistake at the time of recognition process. See the topic "Text Entity Editing Commands/Command: %Join" for details.

Command **%Split** or key **F5** allows splitting of text entity apart, if two or more words were combined by mistake into one text entity at the time of recognition process. See the topic "Text Entity Editing Commands/Command: %Split" for details.

Command **%Del** (Delete) removes the highlighted text entity. Note that you may invoke this command also if you press space bar without clicking anything in the highlighted text entity (otherwise the insertion mark or selection frame will appear, and space will be considered as the replacement character).

Command **%Rid** or key **F6** allows removing of distortion objects, which are located inside of the highlighting frame or too close to it, so it is difficult to select them without selecting of the text entity as well. See the topic "Text Entity Editing Commands/Command: %Rid" for details.

Command **%Obj** or key **F7** allows inserting of new objects (text, polylines, solids, circles). See the topic "General Editing Operations/Operation "Obj" for details.

Key **F8** or clicking the triangle part of the highlighting frame by left mouse button changes text entity to the opposite direction, so it flips upside-down.

Command **%Magn** (Magnify) or key **F9** changes the viewing scale to see the highlighted text entity with the maximum resolution. Note that you may undo it like other commands and return to the original view.

Command **%Zoom** (available only in Selective Mode) invokes AutoCAD command "Realtime Zoom". Note that after closing of that command, program loses focus on the previously highlighted text entity, so you need to click it again.

Command **%vieW_done** or key **F10** (available only in Segment Mode) shows the whole drawing with "grid" on the segments, which were already passed, and with highlighting of already checked or edited objects. It shows also locations of the current segment and the currently highlighted text entity.

Command **%View_seg** (available only in Segment Mode) returns the view to the current segment.

Command **%bacK** (available only in Segment Mode) allows returning to the previous text entity.

Command **%couNt** (available only in Segment Mode) displays the statistic information regarding the number of text entities on the current segment and which part of them is already checked or edited.

Selection **<mark>** may be applied when you want to complete editing of the highlighted text entity. If you press Enter or right mouse button, this text entity will be marked with special "Highlighting Color" defined in Recognition Result Editor Parameters dialog box (see the topic "Recognition Result Editor Parameters"). Marking of checked/edited text entities helps to distinguish them from ones, which are not checked yet.
If you want to leave text entity unmarked, do not press Enter or right mouse button when editing is complete, but press Esc instead (note that Esc does not undo result of editing because it may be undone only by command %Undo or Tab key). The same effect will take place, if you click some another recognized object (text, polyline, solid or circle), so editing focus will be changed.

Editing of Polylines

To make polyline available for editing, it is necessary to click it by left mouse button.
After that the selected polyline will be highlighted, polyline vertices and middle points of arc-segments will be marked by square boxes, and the following message will appear in the command line.

[POLY]Undo/Follow/Arc/Line/Seg_move/Break/Gap/Turn/Prol/Ints/Width/Del/Erase/Rid/Magn/Class/ Help/<mark>
(in Selective Mode)

or

[POLY]Undo/Follow/Arc/Line/Seg_move/Break/Gap/Turn/Prol/Ints/Width/Del/Erase/Rid/Magn/Class/ View_seg/Help/<mark>
(in Segment Mode)

There are three different ways to edit polyline:
1 With direct editing of polyline vertices;
2 With dialog box "Vertex Control";
3 With commands listed in the command line.

Direct editing of polyline vertices
Move Vertex
To move existing polyline vertex:
    1) Click the vertex by left mouse button (all vertices become marked by X-marker after that).
    2) Drag it to the desired position.
    3) Click a new vertex location by left mouse button.

Insert Vertex
To insert a new polyline vertex:
    1) Click by left mouse button the desired place between existing vertices.
    2) Drag cursor to the required position of the inserted vertex.
    3) Click by left mouse button this location.

Delete Vertex
To delete polyline vertex:
    1) Click the vertex.
    2) Use any of the following options:
      - press space bar, or
      - invoke Vertex Control dialog box and then use action "Delete", or
      - press right mouse button, if action "Delete" in Vertex Control dialog box is defined as the default action.
In this case the command line will have the following message:
[MOVE VERTEX]Left mouse button(new location)/Vertex control/<Delete>

New Bulge
To change bulge of a whole arc:
    1) Click the marker in the middle of the arc ("Arc-cursor" will appear).
    2) Change bulge of "Arc-cursor" by moving mouse.
    3) Press left mouse button when you reach the desired bulge.

To change bulge only for a part of arc:
    1) Click the part of arc where the bulge should be changed.
    2) Change bulge of "Arc-cursor" by moving mouse.
    3) Press left mouse button when you reach the desired bulge.

Editing with dialog box "Vertex Control"
Dialog box "Vertex Control" is a special editing tool, which allows you to change current SNAP or MOVE options, or to perform some actions with the selected vertex and/or with adjacent polyline segment(s). To invoke this dialog box you need first to click polyline vertex by left mouse button. Then, if Vertex Control is the default selection (it is written in command line as **<Vertex control>** in this case), you may press Enter or right mouse button, or type "**v**". If Vertex Control is not the default selection (see above sample of

command line for **Delete Vertex**), you need to type "**v**". See the topic "Polyline Editing Commands/Dialog Box "Vertex Control" for information how to use this editing tool.

Each command is invoked by typing of character, which is capitalized in the name of the command (for instance character "g" in the command "Gap"), and, for some commands, by pressing Enter or right mouse button additionally. If pressing of Enter or right mouse button is required, you will see the typed character in the command line, otherwise command starts immediately after you have typed the first character.

Command **Undo** undoes result of the last editing operation.

Command **Follow** invokes special polyline "Follow" mode, which allows tracing of the selected polyline along its entire length, with possibility to edit this polyline or surrounding objects if necessary. See the topic "Polyline Editing Commands/Polyline "Follow" Mode" for details.

Command **Arc** replaces selected polyline segment with one arc-segment by assigning bulge value to the corresponding vertex. See the topic "Polyline Editing Commands/Command: Arc" for details.

Command **Line** allows replacing of some polyline part, which contains several vertices, with one straight two-vertex segment. See the topic "Polyline Editing Commands/Command: Line" for details.

Command **Seg_move** allows parallel moving of selected segment together with all adjacent segments. See the topic "Polyline Editing Commands/Command: Seg_move" for details.

Command **Break** breaks polyline apart at the point, where you click it by left mouse button. See the topic "Polyline Editing Commands/Command: Break" for details.

Command **Gap** allows closing of an open end of polyline on the same or different polyline (in the second case both polylines are joined into one). This feature is usually applied when some raster line should be recognized as one polyline, but it is broken because of gap on raster. See the topic "Polyline Editing Commands/Command: Gap" for details.

Command **Turn** allows changing of polyline direction in an intersection with another polyline. See the topic "Polyline Editing Commands/Command: Turn" for details.

Command **Prol** (Prolong) extends an open end of polyline with or without adding of additional vertices. See the topic "Polyline Editing Commands/Command: Prol" for details.

Command **Ints** (Intersect) allows intersecting of open ends of surrounding polylines with highlighted one (ends will be extended or trimmed). See the topic "Polyline Editing Commands/Command: Ints" for details.

Command **Width** allows changing of width of the whole polyline or of some selected segments. New width may be applied with different beginning and ending values, and may be also spread evenly along the changed part of the polyline. This command may be used even if polyline was recognized with "Width Filter" turned OFF. See the topic "Polyline Editing Commands/Command: Width" for details.

Command **Del** (Delete) removes the highlighted polyline. Note that you can invoke this command also by pressing space bar while no vertex is clicked.

Command **Erase** may be used to remove some local distortion(s) on the polyline (such as "shells", shoots, false segments in the corners and so on) and also to make polyline segments strictly orthogonal. See the topic "Polyline Editing Commands/Command: Erase" for details.

Command **Rid** allows removing of distortion objects, which are located too close to the polyline, so it is difficult to select them without selecting of the polyline as well. See the topic "Polyline Editing Commands/Command: Rid" for details.

Command **Magn** (Magnify) changes the viewing scale to see the highlighted polyline with the maximum resolution. Note that you may undo it by command Undo and return to the original view.

Command **Class** allows changing of layer/color and output width, assigned to the polyline at the time of recognition. See the topic "Polyline Editing Commands/Command: Class" for details.

Command **View_seg** (available only in Segment Mode) returns the view to the current segment.

Selection **<mark>** may be applied when you want to complete editing of the highlighted polyline. If you press Enter or right mouse button, this polyline will be marked with special "Highlighting Color" defined in Recognition Result Editor Parameters dialog box (see the topic "Recognition Result Editor Parameters"). Such marking helps to distinguish already edited polylines from ones, which are not checked yet. Note that polyline may be marked only if it is entirely visible in the current view (for instance if you applied "Magn" command), otherwise color will not be changed.
If you want to leave polyline unmarked, do not press Enter or right mouse button when editing is complete, but press Esc instead (note that Esc does not undo result of editing because it may be undone only by command Undo). The same effect will take place, if you click some another recognized object (text, polyline, solid or circle), so editing focus will be changed.

Editing of Solids

To make solid available for editing, it is necessary to click it by left mouse button.
After that the body of solid will be temporarily invisible (to see underlying raster), the contour of solid will be highlighted, vertices of contour will be marked by square boxes, and the following message will appear in the command line.

[SOLID]Undo/Part/hoLe/Split/Del/Rid/Magn/Class/Trim_pline/Gap_pline/Help/<mark>
(in Selective Mode)

or

[SOLID]Undo/Part/hoLe/Split/Del/Rid/Magn/Class/Trim_pline/Gap_pline/View_seg/Help/<mark>
(in Segment Mode)

There are three different ways to edit solid:
1 With direct editing of contour-polyline vertices;
2 With dialog box "Vertex Control";
3 With commands listed in the command line.

Direct editing of contour-polyline vertices
Move Vertex
To move existing polyline vertex:
   1) Click the vertex by left mouse button (all vertices become marked by X-marker after that).
   2) Drag it to the desired position.
   3) Click a new vertex location by left mouse button.

Insert Vertex
To insert a new polyline vertex:
   1) Click by left mouse button the desired place between existing vertices.
   2) Drag cursor to the required position of the inserted vertex.
   3) Click by left mouse button this location.

Delete Vertex
To delete polyline vertex:
   1) Click the vertex.
   2) Use any of the following options:
     - press space bar, or
     - invoke Vertex Control dialog box and then use action "Delete", or
     - press right mouse button, if action "Delete" in Vertex Control dialog box is defined as the default action.
In this case the command line will have the following message:
[MOVE VERTEX]Left mouse button(new location)/Vertex control/<Delete>

Editing with dialog box "Vertex Control"
Dialog box "Vertex Control" is a special editing tool, which allows you to change current SNAP or MOVE options, or to perform some actions with the selected vertex and/or with adjacent polyline segment(s). To invoke this dialog box you need first to click polyline vertex by left mouse button. Then, if Vertex Control is the default selection (it is written in command line as **<Vertex control>** in this case), you may press Enter or right mouse button, or type "**v**". If Vertex Control is not the default selection (see above sample of command line for **Delete Vertex**), you need to type "**v**".
See the topic "Polyline Editing Commands/Dialog Box "Vertex Control" for information how to use this editing tool.

Command line commands
Each command is invoked by typing of character, which is capitalized in the name of the command (for instance character "t" in the command "Trim_pline"), and, for some commands, by pressing Enter or right mouse button additionally. If pressing of Enter or right mouse button is required, you will see the typed character in the command line, otherwise command starts immediately after you have typed the first character.

Command **Undo** undoes result of the last editing operation.

Command **Part** allows removing of some part of the solid. See the topic "Solid Editing Commands/Command: Part" for details.

Command **hoLe** allows to change or delete existing hole(s) or to add new ones. See the topic "Solid Editing Commands/Command: hoLe" for details.

Command **Split** allows splitting of highlighted solid into two new single solids. See the topic "Solid Editing Commands/Command: Split" for details.

Command **Del** (Delete) removes the highlighted solid. Note that you can invoke this command also by pressing space bar while no vertex is clicked.

Command **Rid** allows removing of distortion objects, which are located too close to the solid, so it is difficult to select them without selecting of the solid as well. See the topic "Solid Editing Commands/Command: Rid" for details.

Command **Magn** (Magnify) changes the viewing scale to see the highlighted solid with the maximum resolution. Note that you may undo it by command Undo and return to the original view.

Command **Class** allows changing of layer/color setting, assigned to the solid at the time of recognition. See the topic "Solid Editing Commands/Command: Class" for details.

Command **Trim_pline** allows cutting of part of some polyline, which intersects with solid. See the topic "Solid Editing Commands/Command: Trim_pline" for details.

Command **Gap_pline** allows joining of ends of some polyline(s), which intersect(s) with solid. See the topic "Solid Editing Commands/Command: Gap_pline" for details.

Command **View_seg** (available only in Segment Mode) returns the view to the current segment.

Selection **<mark>** may be applied when you want to complete editing of the highlighted solid.
If you press Enter or right mouse button, this solid will be marked with special "Highlighting Color" defined in Recognition Result Editor Parameters dialog box (see the topic "Recognition Result Editor Parameters"). Such marking helps to distinguish already edited solids from ones, which are not checked yet.
If you want to leave solid unmarked, do not press Enter or right mouse button when editing is complete, but press Esc instead (note that Esc does not undo result of editing because it may be undone only by command Undo). The same effect will take place, if you click some another recognized object (text, polyline, solid or circle), so editing focus will be changed.

Operation "Amend Corners"

This operation automatically finds corners between polyline segments on the whole drawing and removes distortion segment(s) in these corners. It provides also orthogonal adjustment, if one side of corner is close to horizontal and another side to vertical direction (unlike polyline editing command "Erase", orthogonal adjustment may be applied even if there is no any distortion segment between corner sides).
The term "amending" means removing of distortion segments and/or orthogonal adjustment of corner sides.
The term "corner" means two adjoining polyline segments with certain angle between them, which is specified by parameters: "Minimum angle between corner sides" and "Maximum angle between corner sides". If angle between segments is in the defined range, these segments are considered as "corner" independently on whether they are orthogonal or with arbitrary directions.

Note: Operation "Amend Corners" may be called only in Selective mode because it starts its own editing process on the whole drawing, so it cannot be invoked in Segment mode.

After operation starts, it first of all counts number of corners for potential amending (on the whole drawing) and displays the following messages in the command line:

Number of corners to amend=xx
Start? /Start/Exit/Parameters/Help/<Start>:

To start amending process, press Enter or right mouse button. After that program shows the first corner and displays the following messages in the command line:

Done=0, Left=xx
Select action: /Amend/Skip/Exit/Zoom/Blind/Help/<Amend>:

If you want to amend the displayed corner, press Enter or right mouse button. This corner will be amended and the following message will appear in the command line:

Is result of amending OK? /Ok/Undo/Zoom/Help/<Ok>:

If you accept result of amending, press Enter or right mouse button, and program will show you the next corner to amend. If you do not like result of amending, type "**u**" and press Enter or right mouse button to undo this result. If you want to change scale of current view, type "**z**" and press Enter or right mouse button to invoke special EasyDone command **Zoom**.

If you do not want to amend some corner, you may use option **Skip** in the "Select action" message. In this case type "**s**" and press Enter or right mouse button. The displayed corner will be skipped, and program will proceed to the next corner.

You may also switch at any moment to the completely automatic mode of amending. In this case use option **Blind** in the "Select action" message (type "**b**" and press Enter or right mouse button).

If you want to change parameters of amending, you may select option **Parameters** in the first message (type "**p**" and press Enter or right mouse button). If you want to change parameters during amending process, you should quit using option **Exit** in the "Select action" message (press Esc or type "**e**" and press Enter or right mouse button) and then start operation again. Note that all previously amended corners will not be shown any more after secondary start of this operation (however all skipped corners will be shown again).

If you use option **Parameters**, the following dialog box will appear.
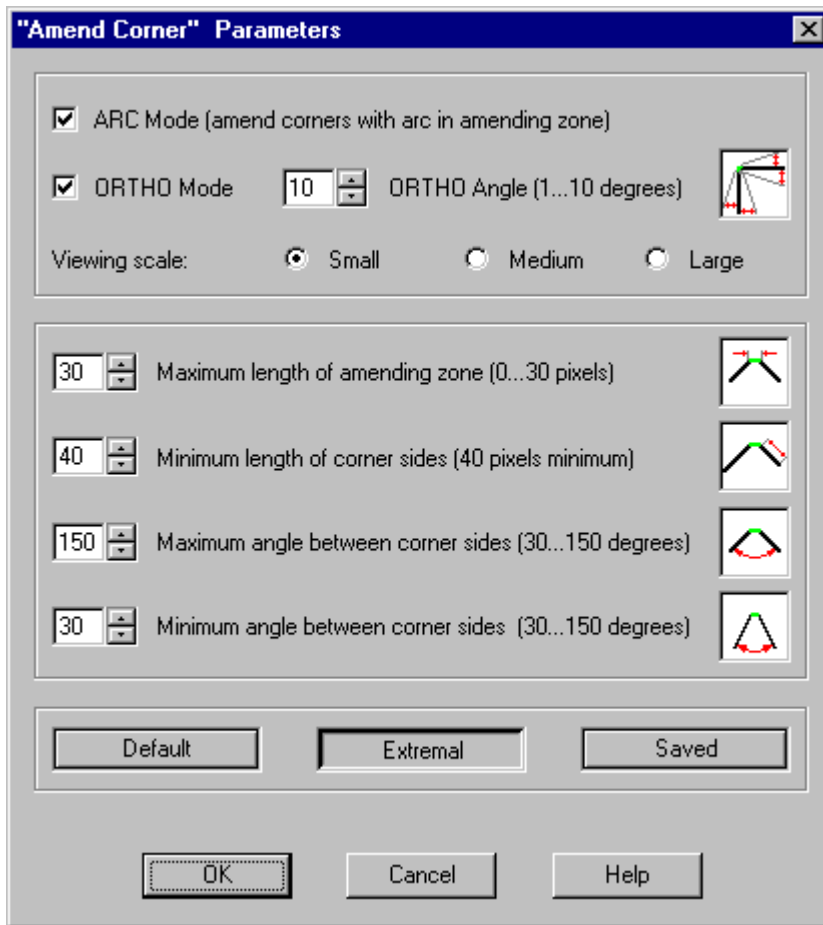
ARC Mode
This option allows removing of distortion segments if some of them have bulge value (i.e. they are arc-segments). If this option is not checked, corners with such segments will be automatically skipped.

ORTHO Mode
This option defines whether program should apply orthogonal adjustment to the corner sides, if both sides do not exceed angle with corresponding axis specified in ORTHO Angle parameter.

ORTHO Angle
This parameter defines maximum angle between corner side and corresponding axis when orthogonal adjustment is possible. If either of corner sides exceeds this angle, orthogonal adjustment will not be applied.

Viewing scale
This parameter defines how detailed each corner will be displayed.

Maximum length of amending zone
This parameter defines maximum distance between adjoining ends of corner sides when amending is possible (if there is no any distortion segment between them, these sides are adjacent and length of the amending zone is zero). If distance exceeds the defined value, amending will not be applied and this corner will be automatically skipped.

Minimum length of corner side
This parameter excludes from amending process corners, which have one or both sides shorter than defined length. It helps to avoid consideration of segments, which are not actually "corners".

Maximum angle between corner sides
This parameter excludes from amending process corners with too obtuse angle between their sides.

Minimum angle between corner sides
This parameter excludes from amending process corners with too sharp angle between their sides.

Button "Default" sets all parameters to the recommended pre-defined values.

Button "Extremal" sets all parameters to their max/min limits.

Button "Saved" restores the last customized setting. When you change some of the amending parameters, this customized setting will be saved automatically (if you close dialog box with OK) and then available by this button even if you pressed earlier button "Default" or "Extremal".

TIP: The current parameter setting corresponds to the button, which is currently highlighted in the dialog box as "pressed".

If you have changed some of amending parameters (except "Viewing scale"), program will count number of corners for potential amending from the beginning.

Note: The counted number may exceed actual number of remaining corners because some corners may become orthogonal automatically when program amends another corners. For example, if there is some rectangle with all sides close to orthogonal directions, the number of corners will be counted as four. But when three of rectangle corners are amended, the forth corner becomes orthogonal too, as result of orthogonal adjustment of other three corners, so it will be skipped by program (if there is no distortion segment in the forth corner, so it requires only orthogonal adjustment).

Operation "Obj" (Add Object)

Note: Operation may work improperly, if image was recognized with UNIT value "Miles" or "Kilometers". Use AutoCAD commands to insert a new entity in this case.

This operation allows inserting of new entities, such as text, polylines, solids or circles, and may be invoked by command **Obj** in Selective or Segment mode, or by command **%Obj** when you edit text (for instance, you may use this command in text editing mode if you do not want edit some recognized text entity but prefer rather to remove it completely and to insert new one instead).

The following message will appear in the command line:

[ADD]Text/Polyline/Solid/Circle/Help/

Text
To insert new text entities:
   - Type "**t**" and "Text Classes" dialog box will appear.
   - Select desired class or press button "Change parameters…" and define a new class.
   - Input insertion point, height, rotation angle, and inserting text.
   - Repeat previous step to insert additional text entities or press Enter or right mouse button to complete operation.

TIP: You may input height and rotation angle either by typing of their values or by mouse clicks.

The inserted text entities are considered as already edited, so they will be automatically marked with "Highlighting Color" defined in Recognition Result Editor Parameters dialog box.

Polyline
To insert new polylines:
   - Type "**p**" and "Polyline Classes" dialog box will appear.
   - Select desired class or press button "Change parameters…" and define a new class.
   - Input all points of inserting polyline and press Enter or right mouse button after that.
   - Repeat previous step to insert additional polylines or press Enter or right mouse button to complete operation.

TIP: If you want to create closed polyline, set option Snap/Move to **Vertex** or **Polyline End** in "Vertex Control" dialog box (because in this case the special triangle marker will show snapping of the first end of inserting polyline) and close the last polyline vertex on the first end.

The inserted polylines are considered as already edited, so they will be automatically marked with "Highlighting Color" defined in Recognition Result Editor Parameters dialog box.

Solid
To insert new solids:
   - Type "**s**" and "Solid Classes" dialog box will appear.
   - Select desired class.
   - Input points of solid contour (except the closing point) and press Enter or right mouse button.
   - Answer whether you want to create hole(s) and input their points, if yes.
   - Repeat previous step to insert additional solids or press Enter or right mouse button to complete operation.

The inserted solids are considered as already edited, so they will be automatically marked with "Highlighting Color" defined in Recognition Result Editor Parameters dialog box.

Circle
To insert new circles:
   - Type "**c**" and "Polyline Classes" dialog box will appear.
   - Select desired class or press button "Change parameters…" and define a new class.
   - Input two points on diameter of inserting circle and press Enter or right mouse button.
   - Repeat previous step to insert additional circles or press Enter or right mouse button to complete operation.

Note: When you select class with non-zero polyline width, new circle will be inserted in the form of polyline, otherwise in the form of AutoCAD CIRCLE entity.

TIP: If you want to insert circle as polyline but with zero width, select class with non-zero width and then change width of inserted polyline-circle to zero with polyline editing command "Width".

The inserted circles are considered as already edited, so they will be automatically marked with "Highlighting Color" defined in Recognition Result Editor Parameters dialog box.

Operation "String"

This operation allows joining of text entities into one string with keeping of spaces between these text entities (unlike text editing command %Join). You may use this operation to join into one string single words as well as another strings that may be useful if you cannot select all required text entities in the current view at once.

After command starts, the following messages appear in the command line.

Select starting word or substring to create new string: Esc/Select/Help/<Select>
Press Enter or right mouse button to proceed.

Click text entity:
Click by left mouse button the first (leftmost) text entity in the creating string.

Pick corner of the frame around joining text entities #1/Help/
Click by left mouse button the first corner of the selection frame around other joining text entities (in any order).

Pick corner of the frame around joining text entities #2/Help/
Click by left mouse button the second corner of the selection frame (in any order).

Pick corner of the frame around joining text entities #3/Help/
Click by left mouse button the third corner of the selection frame (in any order).

After that the new string will be created and its left and right points will be marked by X-marker.
The following message will appear in the command line.

Is string OK? /Undo/Ok/Add_space/Delete_space/Help/<Ok>:
If created string does not require additional correction, press Enter or right mouse button to complete operation.
If you want to undo result of string creation, type "u" and press Enter or right mouse button.
If you want to add additional space between joined text entities type "**a**" (option **Add_space**) and press Enter or right mouse button. One space will be added to each text entity in the created string. To add more spaces, repeat this operation.
If you want to remove space between joined text entities type "**d**" (option **Delete_space**) and press Enter or right mouse button. One space will be removed from each text entity in the created string. To remove more spaces, repeat this operation.

TIP: If joined text entities were recognized with option "Automatic Width" turned ON (see the topic "Text Recognition Parameters"), then operations **Add_space** and **Delete_space** will change width of each joined text entity, but the right insertion point of the whole string will not be changed. If they were recognized with option "Automatic Width" turned OFF, then operations **Add_space** and **Delete_space** will not change width of each text entity, but will change position of the right insertion point of the string.

Dialog Box "Text Classes"

Note: This feature may work improperly with AutoCAD 2005 because of bug in AutoCAD 2005.

Dialog box "Text Classes" may be called by command %Class for changing of text entity attributes assigned at the time of recognition, or for specifying of these attributes for operation Add Object (see the topic "Operation "Obj"").

The following attributes are specifically related to each class: pixel height limits, font height, font oblique angle, font width, text style, layer, and color. These attributes are taken from the current parameter setting, defined in Text Recognition Parameters dialog box.
If you want to change this setting, press button "Change parameters". After that Text Recognition Parameters dialog box will appear, where you may change desired attributes for each class, or to define a new class. Note that some parameters will be disabled because they are not related to class attributes. Parameters "Mode" and "Height Limits" will be available, but they do not affect class attributes within editing process, so it is not recommended to change these parameters.

Automatic Width
This option may be changed after pressing of button "Change parameters". It may be helpful in the following two cases:
- Text entity was recognized with Automatic Width turned OFF, and you cannot change width of this text entity with frame-based operation because of that (see the topic "Editing of Text").
- Text entity was recognized with Automatic Width turned ON, and attribute Font Width is not available because of that.

Button "Restore original set" becomes available after you used button "Change parameters" and changed some of parameters in Text Recognition Parameters dialog box. You may use this button to return changed parameters to their original values, if you decided to discard changes.
You may also use changed parameters for command %Class (you need to close dialog box for that) and then call command %Class again and press button "Restore original set", however it is not necessary if you use option "Restore original set when quit editing mode".

Check box "Restore original set when quit editing mode" becomes available after you used button "Change parameters" and changed some of parameters in Text Recognition Parameters dialog box. If this option is turned ON, all changed parameters will be restored to their original values after you quit Recognition Result Editor. It is recommended to keep this option always turned ON.

If dialog box "Text Classes" is called by command %Class, there are two possible situations:
1. Text entity, for which command %Class is used, corresponds by all attributes to some class. In this case the radio button of this class will be checked when dialog box is opened. To change class, just check radio button of the desired class.

2. Text entity does not correspond exactly to any of currently defined classes. In this case no one of radio buttons will be checked when dialog box is opened. To set attributes of desired class check radio button of that class.

If dialog box "Text Classes" is called by operation "Obj" (Add Object), no radio buttons will be checked when dialog box is opened. To set attributes of desired class check radio button of that class.

Command: %Join (text editing)

This command allows joining of the highlighted text entity with another text entity, from which it was separated by mistake at the time of recognition process.

After command %Join is started, you need to press Enter or right mouse button and then to click by left mouse button the text entity, which should be combined with the highlighted one.

TIP: Do not use this command when you want to combine single words into one string, because these words will be combined without spaces between them. Use operation "String" for this purposes (see the topic "Operation "String").

Command: %Split (text editing)

This command allows splitting of the highlighted text entity apart, if two or more words were combined by mistake into one text entity at the time of recognition process.

After command %Split is started, you need to click by left mouse button the first character in the part, which you want to split (in the command line message it is called "second" part).
Then you may pick by left mouse button new insertion point for this part, or to press Enter or right mouse button and to leave this part at its current position.

After that you may pick by left mouse button right insertion point for the "first" part, or to press Enter or right mouse button and to leave this part at its current position.
TIP: If text entity was recognized with option "Automatic Width" (see the topic "Text Recognition Parameters") turned OFF, this step will be skipped because right insertion point is absent.

When command %Split is complete, the "first" part gets focus and becomes currently highlighted text entity.

Command: %Rid (text editing)

This command allows removing of distortion objects, which are located inside of the highlighting frame or too close to it, so it is difficult to select them without selecting of the text entity as well. Even if text entity is selected along with distortion objects, it will not be deleted due to this command.

After command %Rid is started, you need to press Enter or right mouse button and then to select objects, which you want to delete. When selecting is done, press Enter or right mouse button again to complete operation.

TIP: If distortion object is solid with hole(s), you should make selection so, that all holes are selected, otherwise the contour of solid may be deleted but hole(s) remain.

Dialog Box "Vertex Control" (polyline editing)

To invoke Vertex Control dialog box for editing of some vertex and/or adjacent segment(s), click this vertex by left mouse button and then press Enter or right button (if **<Vertex Control>** is the default selection in the command line) or type "**v**". The following dialog box will appear.

TIP: Vertex Control commands may be used every time when "Vertex Control" option is listed in the command line.

Snap/Move Group

These options specify which vertex to snap, and allow restriction of mouse cursor movement to certain directions.

Vertex

This option allows snapping of any vertex on the highlighted or another polyline.

Nearest

Select this option to snap a point, precisely located on a polyline or circle, which is closer to the current position of mouse cursor.

Intersection

Select this option to snap the precise point of intersection of polylines, polyline(s) and circle(s), circle(s) and circle(s).

Polyline End

Select this option to snap vertex located in a polyline end.

TIP: When mouse cursor reaches the selected point of snap, the special triangle marker appears to highlight this point and to inform you that snapping is available.

Ortho

Select this option to restrict mouse cursor movement from one point to another precisely along the horizontal or vertical direction.

Direction

Select this option to restrict mouse cursor movement from one point to another precisely along some selected direction.
You will be prompted to define direction using any of the following options:
- press right mouse button on a polyline segment (end points of this segment will define desired direction);
- click two points;
- input angle value.

TIP: Options "Ortho" and "Direction" may be useful when you need to move or copy something in parallel to its original position. It is recommended to use these options just temporarily, when you really need it, and to turn them OFF after using, otherwise you may forget that you have turned such option ON, and will be confused later why mouse cursor does not move normally.

Off
Select this option to disable other selections of Snap/Move group.

Together
When this option is ON and some intersection point is moved to a new location, the adjacent segments of all polylines, passing through this intersection, will move to that location too.

Set as Default
If this option is ON, the current Snap/Move parameter is used as default every time when "Vertex control" option is presented in the command line.

Action Group
This group provides the following operations after some vertex is clicked.

Intersect
If selected vertex is a polyline end, the adjacent segment will be extended or trimmed to intersect the nearest another polyline or circle.
To avoid possible mistakes, the intersection point must be visible on the screen.

Vertical
This function makes strictly vertical the segment, adjacent to the selected vertex, by moving this vertex along the horizontal direction.
When selected vertex has two adjacent segments:
a) if one segment is closer to vertical and another to horizontal direction, the first one will be adjusted to vertical direction;
b) otherwise you will be prompted to select one of these segments to make it vertical.

Horizontal
This function makes strictly horizontal the segment, adjacent to selected vertex, by moving this vertex along the vertical direction.
When selected vertex has two adjacent segments:
a) if one segment is closer to horizontal and another to vertical direction, the first one will be adjusted to horizontal direction;
b) otherwise you will be prompted to select one of these segments to make it horizontal.

Corner
When selected vertex has two adjacent segments, one segment is closer to horizontal, and another segment is closer to vertical direction, the selected vertex will be moved to make the first segment strictly horizontal and the second segment strictly vertical.

Delete
This function deletes the selected vertex.

If you want to define some action "by default", you may click the corresponding radio button. In this case the name of action will appear as default selection in the command line and may be invoked by pressing Enter or right mouse button without displaying of "Vertex Control" dialog box.

Default Off
Use this option to disable current "by default" selection.

Polyline "Follow" Mode

Polyline "follow" mode is a special mode of polyline editing, in which Recognition Result Editor moves along the selected polyline screen by screen. This mode is useful for long polylines, which cannot fit in one screen with scale sufficient for seeing of all necessary details. It may be used, for instance, with different GIS drawings (contour maps, soil maps, parcel maps) and other drawings with long graphical objects.

In this mode you may:
- use any command to edit the selected polyline like in regular **[POLY]** mode (see the topic "Editing of Polylines");
- click and edit any other objects (text, polylines, solids, circles) and then continue "follow" mode.

Type "**f**" to start "follow" mode (after you clicked some polyline in Selective or Segment mode).
The following message will appear in the command line:

[FOLLOW]Undo/reVerse/Arc/Line/Seg_move/Break/Gap/Turn/Prol/Ints/Width/Del/Erase/Rid/Magn/ Class/Help/<ok>

Markers "arrow-head" and "arrow-tail" will appear on the currently visible part of polyline to indicate the direction of moving.

There are three specific options in addition to other polyline editing commands:

reVerse
This option may be invoked by typing "**v**" and changes direction of moving along the polyline.
Moving will continue from the next screen, adjacent to the point of start, but in the opposite direction.

<ok>
To mark the currently visible part of polyline and move to the next screen, press right mouse button or Enter, or type "**f**".

When the first end of polyline is reached, Recognition Result Editor automatically performs command "Reverse" (if "follow" mode has been started in the middle of polyline).

<back>
This may option appear instead of **<ok>** if the "arrow-tail" is not visible on the current screen (for instance if you changed current view by scroll-bars). Press Enter or right mouse button to return to the original view in this case.

When the "follow" mode is complete (the whole polyline was traced), Recognition Result Editor automatically returns to the point of start. Press Esc if you want to abort "follow" mode before that.

Note: When you click within "follow" mode some another object to edit it, the polyline will be temporarily highlighted with "Segment Frame Color", defined in Recognition Result Editor Parameters dialog box (see the topic "Recognition Result Editor Parameters"). After editing of clicked object is completed or aborted, the control returns to the "follow" mode and original highlighting is restored.

Command: Arc (polyline editing)

To substitute some part of polyline between any two points (not necessarily located in vertices) by arc-segment:

1) Click the first point on polyline.
2) Click the second point on polyline ("arc cursor" will appear).
3) Change bulge of "arc cursor" by moving mouse.
4) Press left mouse button when you get desired bulge.

On steps 1) and 2) you may press Enter or right mouse button to invoke Vertex Control dialog box, which allows to turn on SNAP mode with options: "Vertex", "Intersection", "Polyline End" (see the topic "Dialog Box "Vertex Control"). When command starts, the SNAP mode is turned off.

Command: Line (polyline editing)

To substitute polyline part between any two points (not necessarily located in vertices) by a straight segment:
   1) Click the first point on polyline.
   2) Click the second point on polyline.

On these steps you may press Enter or right mouse button to invoke Vertex Control dialog box, which allows to turn on SNAP mode with options: "Vertex", "Intersection", "Polyline End" (see the topic "Dialog Box "Vertex Control"). When command starts, the SNAP mode is turned off.

Command: Seg_move (polyline editing)

This command allows moving of polyline segment along with adjacent segment(s) in parallel to its original position. It may be helpful if some polyline segment moved away from the original raster line because of orthogonal adjustment or another reason. Using of this command instead of independent moving of segment vertices allows keeping of exact segment orientation, that is important, for instance, if segment was adjusted to orthogonal direction, etc.

To move segment of the highlighted polyline:
  1) Click this segment by left mouse button.
  2) Select direction to move segment (any, vertical, horizontal).
  3) Pick the displacement point.

To select direction on step 2) you may press Enter or right mouse button for default selection, or to type character "**a**", "**v**", or "**h**" correspondingly and press Enter or right mouse button.
If you select vertical direction, then only vertical component of displacement will be used.
If you select horizontal direction, then only horizontal component of displacement will be used.
If you select any direction, then both components of displacement will be used.
TIP: The default selection (any, vertical, or horizontal) is defined automatically depending on whether selected segment is closer to vertical or horizontal axis, or neither (case "Any").

Command: Break (polyline editing)

To break polyline apart click by left mouse button the desired break point (not necessarily located in a vertex).

You may press Enter or right mouse button to invoke Vertex Control dialog box, which allows to turn on SNAP mode with options: "Vertex", "Intersection", "Polyline End" (see the topic "Dialog Box "Vertex Control"). When command starts, the SNAP mode is turned off.

If the original polyline is open, the first resulting polyline will be created between the first end and the break point, and the second resulting polyline will be created between the second end and the break point.

If the original polyline is closed, both resulting polylines will be created between starting/ending vertex and the break point, but on different parts of the original polyline.

Command: Gap (polyline editing)

To create link (consisted of one or more segments) through a gap between open end of the highlighted polyline and open end of another polyline (or between two open ends of highlighted polyline):

    1) Click the first polyline end (it is not necessary to click exactly the end and even the polyline itself - the end closest to the point of click will be chosen automatically).

    2) Click one by one the locations of intermediate vertices (if inserting link should have more than one segment).

    3) Locate cursor at the end of the second polyline and press right mouse button (it is not necessary to locate cursor exactly at the end, just on the polyline - the closest end will be chosen automatically).

If link was created between the ends of two different polylines, both polylines will be automatically joined into one entity with layer name and color of the first polyline.

In polyline "follow" mode step 1) will be skipped - the end marked with "arrow-head" will be chosen automatically.

Command: Turn (polyline editing)

To change direction of the highlighted polyline in an intersection with another polyline:
   1) Click another polyline to make a turn (if this polyline intersects the highlighted polyline in more then one point, the point closest to the point of click will be chosen automatically).
   2) Click the part of highlighted polyline to keep editing (if intersection point splits the highlighted polyline into two parts).

In polyline "follow" mode step 2) will be skipped - the part, adjacent to the beginning of highlighted polyline, will be chosen automatically.

Command: Prol (Prolong) (polyline editing)

To prolong the highlighted polyline beyond one of its ends:
   1) Click the polyline end to be prolonged (it is not necessary to click exactly the end and even polyline itself - the end closest to the point of click will be chosen automatically).
   2) Click location of the next point to create additional segment, or type "**b**" to change bulge of the previous segment, or type "**v**" to invoke Vertex Control dialog box.
   3) Repeat step 2) or press Enter or right mouse button to complete operation.

Vertex Control dialog box allows to turn on SNAP/MOVE mode with options:
"Vertex", "Nearest", "Intersection", "Polyline End", "Ortho", "Direction" (see the topic "Dialog Box "Vertex Control").

Command: Ints (Intersect) (polyline editing)

To extend or to cut the end of some polyline to the intersection with highlighted polyline:
   1) Click desired end of the polyline to be extended/cut (it is not necessary to click exactly the end, just the polyline - the end closest to the point of click will be chosen automatically).
   2) Repeat step 1) with another polyline(s) or press Enter or right mouse button to complete operation.

To avoid possible mistakes, the end of polyline will be moved only if point of intersection with highlighted polyline is visible on the screen.

Command: Width (polyline editing)

This command allows changing of width of the whole polyline or only selected segment(s).
Width may be changed with different starting and ending values, if necessary. You may also spread width evenly along selected segments or the whole polyline, if starting and ending width are different.

To use this command, follow questions and instructions in the command line (note that if you want to change width of just one segment, you should click it twice when prompted).

TIP: You may use dialog box "Polyline Classes" to specify desired new width, or define it by typing new width value, or by two mouse clicks. If you use dialog box "Polyline Classes", then only width parameter of selected class will be used (layer and color of polyline will not be changed).

Command: Erase (polyline editing)

This command allows erasing of distortion polyline segments, which were created at the time of recognition as result of local distortions on raster line. This command provides also orthogonal adjustment of polyline segments, one of which is close to vertical and another to horizontal direction, in the case if there is distortion segment(s) between them.
TIP: Use dialog box "Vertex Control" for orthogonal adjustment if there is no distortion segment. Note that such segments may be also adjusted to strictly orthogonal position within operation "Amend Corners" (see the topic "Operation "Amend Corners").

The following message appears in the command line after command "Erase" starts:

Select rectangular area around erasing segment(s): Esc/Ortho/Select/Help/<Select>

To proceed to the next step and select segment(s), which should be erased, press Enter or right mouse button and then click by left mouse button two corners of rectangular area around erasing segments. When this area is selected, press left mouse button to complete operation.
TIP: If selection frame captures some other objects, it does not matter because only segments of the highlighted polyline participate in this operation.

If you want to specify parameters of orthogonal adjustment, type "**o**" (for option **Ortho**) and press Enter or right mouse button. The following message will appear in the command line:

You may turn ORTHO mode Off or change angle limit=xx: Off/Limit/Help/<Esc>

Parameter **angle limit** defines the maximum angle between segment, which may be adjusted to strictly orthogonal position, and corresponding axis. If such angle of either of two segments (horizontal and vertical) exceeds the defined limit, orthogonal adjustment will not be applied.
To change current value of angle limit, type "**l**" (for option **Limit**) and press Enter or right mouse button, and then type new angle value.
If you want to turn orthogonal adjustment OFF, type "**o**" and press Enter or right mouse button.
To check changes made in parameters of orthogonal adjustment, use option **Ortho** in the first message again.
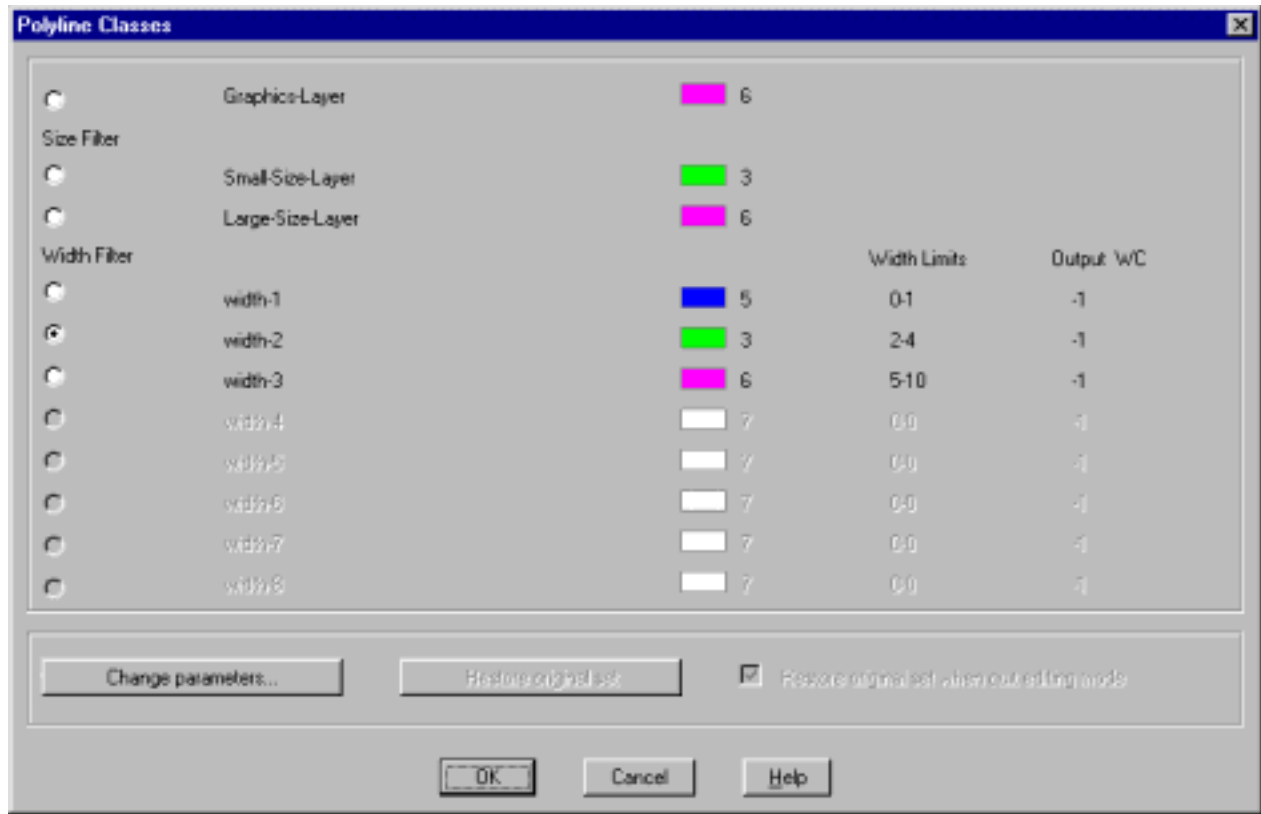
Command: Rid (polyline editing)

This command allows removing of distortion objects, which are located too close to the highlighted polyline, so it is difficult to select them without selecting of the polyline as well. Even if polyline is selected along with distortion objects, it will not be deleted due to this command.

After command "Rid" is started, you need to press Enter or right mouse button and then to select objects, which you want to delete. When selecting is done, press Enter or right mouse button again to complete operation.

TIP: If distortion object is solid with hole(s), you should make selection so, that all holes are selected, otherwise the contour of solid may be deleted but hole(s) remain.

Dialog Box "Polyline Classes"

Note: This feature may work improperly with AutoCAD 2005 because of bug in AutoCAD 2005.

Dialog box "Polyline Classes" may be called by polyline editing command "Class" for changing of polyline attributes assigned at the time of recognition, or for specifying of these attributes for operation Add Object (see the topic "Operation "Obj").

The following attributes are specifically related to each class: layer, color, pixel width limits, output WC (last two only for "Width Filter" sub-group). These attributes are taken from the current parameter setting, defined in Graphics Recognition Parameters, Graphics Size Filter Parameters and Graphics Width Filter Parameters dialog boxes.
If you want to change this setting for "Width Filter" sub-group, press button "Change parameters". After that Graphics Width Filter Parameters dialog box will appear, where you may change desired attributes for each class, or to define a new class. Note that some parameters will be disabled because they are not related to class attributes. Parameters "Width Limits" will be available, but they do not affect class attributes within editing process, so it is not recommended to change these parameters.
If you want to change attributes defined in Graphics Recognition Parameters or Graphics Size Filter Parameters dialog boxes, you need to quit Recognition Result Editor and change parameters directly in those boxes.

Button "Restore original set" becomes available after you used button "Change parameters" and changed some of parameters in Graphics Width Filter Parameters dialog box. You may use this button to return changed parameters to their original values, if you decided to discard changes.
You may also use changed parameters for command "Class" (you need to close dialog box for that) and then call command "Class" again and press button "Restore original set", however it is not necessary if you use option "Restore original set when quit editing mode".

Check box "Restore original set when quit editing mode" becomes available after you used button "Change parameters" and changed some of parameters in Graphics Width Filter Parameters dialog box. If this option is turned ON, all changed parameters will be restored to their original values after you quit Recognition Result Editor. It is recommended to keep this option always turned ON.

If dialog box "Polyline Classes" is called by command "Class", there are two possible situations:

1. Polyline, for which command "Class" is used, corresponds by all attributes to some class. In this case the radio button of this class will be checked when dialog box is opened. To change class, just check radio button of the desired class.

2. Polyline does not correspond exactly to any of currently defined classes. In this case no one of radio buttons will be checked when dialog box is opened. To set attributes of desired class check radio button of that class.

If dialog box "Polyline Classes" is called by operation "Obj" (Add Object), no radio buttons will be checked when dialog box is opened. To set attributes of desired class check radio button of that class.

Note: If image was recognized with UNIT value "Miles" or "Kilometers", then using of automatic setting for attribute "Output WC" (value "-1", see the topic "Graphics Width Filter Parameters") will change width of polyline to zero. In this case it is recommended to use AutoCAD commands for changing width rather than command "Class".

Command: Part (solid editing)

To change/delete part of contour of the highlighted solid between any two points on this contour (not necessarily located in the contour vertices):
1) Click the first point of the new part of contour.
2) Click next point of the new part of contour.
3) Repeat step 2) or press Enter or right mouse button to complete input of new part of contour (only if the last point was clicked on the contour of solid).
4) Click old part of contour to delete.

On steps 1) and 2) you may press Enter or right mouse button, or type "**v**" to invoke Vertex Control dialog box.
Vertex Control dialog box allows to turn on SNAP/MOVE mode with options:
"Vertex", "Nearest", "Intersection", "Polyline End", "Ortho", "Direction" (see the topic "Dialog Box "Vertex Control").
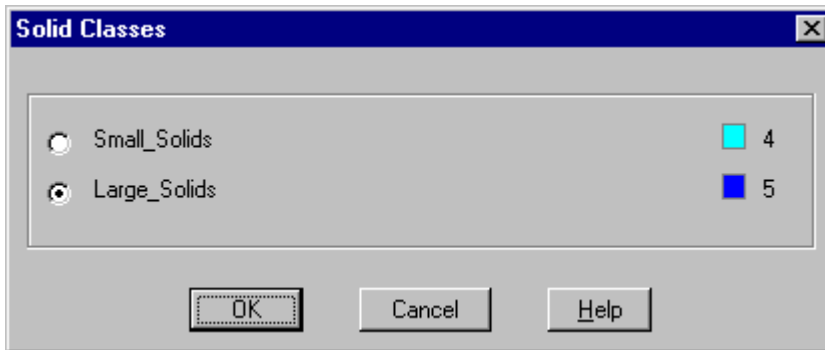
Command: Hole (solid editing)

This command allows to add, to delete, or to change the contour of some hole.
Options "Add", "Delete", "Change" appear in the command line after command starts.

Add
To add a hole inside the highlighted solid:
1) Type "**a**".
2) Click the location of the first point of the hole contour.
3) Click the location of the next point of the hole contour.
4) Repeat step 3) or press Enter or right mouse button to complete operation.

On steps 2) and 3) you may press Enter or right mouse button, or type "**v**" to invoke Vertex Control dialog box.
Vertex Control dialog box allows to turn on SNAP/MOVE mode with options:
"Vertex", "Nearest", "Intersection", "Polyline End", "Ortho", "Direction" (see the topic "Dialog Box "Vertex Control").

Delete
To delete hole from highlighted solid:
1) Type "**d**".
2) Click contour of hole to delete this hole.

Change
To change the contour of hole:
1) Type "**c**".
2) Use the same technique as for editing of solid contour (see the topic "Editing of Solids").

Command: Split (solid editing)

To split highlighted solid into two single solids:
1) Click location of the first point on the contour of solid.
2) Click location of the next point of boundary between splitting parts.
3) Repeat step 2) or press Enter or right mouse button to complete operation (only if the last point was clicked on the contour of solid).
4) Click a part of the old contour to select one of new solids to continue editing.

On steps 1) and 2) you may press Enter or right mouse button, or type "**v**" to invoke Vertex Control dialog box.
Vertex Control dialog box allows to turn on SNAP/MOVE mode with options:
"Vertex", "Nearest", "Intersection", "Polyline End", "Ortho", "Direction" (see the topic "Dialog Box "Vertex Control").

Command: Rid (solid editing)

This command allows removing of distortion objects, which are located too close to the highlighted solid, so it is difficult to select them without selecting of this solid as well. Even if solid is selected along with distortion objects, it will not be deleted due to this command.

After command "Rid" is started, you need to press Enter or right mouse button and then to select objects, which you want to delete. When selecting is done, press Enter or right mouse button again to complete operation.

TIP: If distortion object is another solid with hole(s), you should make selection so, that all holes are selected, otherwise the contour of solid may be deleted but hole(s) remain.

Dialog Box "Solid Classes"

Note: This feature may work improperly with AutoCAD 2005 because of bug in AutoCAD 2005.

Dialog box "Solid Classes" may be called by solid editing command "Class" for changing of solid attributes assigned at the time of recognition, or for specifying of these attributes for operation Add Object (see the topic "Operation "Obj").

The following attributes are specifically related to each class: layer and color. These attributes are taken from the current parameter setting, defined in Solid Recognition Parameters dialog box.
If you want to change these attributes, you need to quit Recognition Result Editor and change parameters directly in that dialog box.

If dialog box "Solid Classes" is called by command "Class", there are two possible situations:
1. Solid, for which command "Class" is used, corresponds by layer and color to one of classes.   In this case the radio button of this class will be checked when dialog box is opened. To change class, just check radio button of another class.
2. Solid does not correspond by layer and color to any of currently defined classes. In this case no one of radio buttons will be checked when dialog box is opened. To set attributes of desired class check radio button of that class.

If dialog box "Solid Classes" is called by operation "Obj" (Add Object), no radio buttons will be checked when dialog box is opened. To set attributes of desired class check radio button of that class.

TIP: If you changed class of recognized solid and also made some other changes to that solid, for instance changed contour of solid or holes, this solid will keep the color defined by new class and will not be highlighted with "Highlighting Color" (see the topic "Recognition Result Editor Parameters"). It is necessary to distinguish such solid from ones, which were modified without changing of their classes.

Command: Trim_pline (solid editing)

This command allows trimming of polyline intersecting (merging) solid.
To trim such polyline just click the part of polyline, which should be deleted, and this part will be trimmed by the contour of solid (note that holes cannot be used for this operation).

Command: Gap_pline (solid editing)

To create polyline link (consisted of one or more segments) through the gap between ends of polyline(s):
1) Click the end of the first polyline (it is not necessary to click exactly the end, just polyline itself - the end closest to the point of click will be chosen automatically).
2) Click one by one locations of the intermediate vertices (if desired link should have more than one segment).
3) Locate cursor on the end of the second polyline (or on the second end of the same polyline) and press right mouse button (it is not necessary to locate exactly on the end, just on the polyline because the closest end will be chosen automatically).

If the link is created between the ends of two polylines, both polylines will be joined into one entity with layer name and color of the first polyline.

This command works the same way as command "Gap" in polyline editing mode and is included in solid editing commands for convenience, to avoid switching from solid to polyline editing mode.

Raster Operations: Menu "Raster Image"

Save
This command saves raster image, if it was modified, along with current DWG file. If raster image was not modified, program will notify you that command "Save" cannot be applied and you should use command "Save As" instead. Raster image is saved with its original name and format (for instance TIFF, PCX, etc.).

Save As
This command saves raster image along with current DWG file independently on whether raster image was modified or not. Raster image is saved with name and format, selected by user.
Note: If name or format of raster image differs from the original, the previously attached image will be detached from DWG file, and the saved image attached instead.
Command "Save As" saves image only in formats, which are supported by AutoCAD. If you want to save image in some format not supported by AutoCAD, you may use command "Export".

Important Notice: If you modified raster image with EasyDone raster editing commands, you should use either EasyDone command "Save" or "Save As" before you save current drawing with AutoCAD's command "Save" or "Save As", otherwise raster changes will not be actually saved.
AutoCAD's commands "Save" and "Save As" save only drawing file and, even if changes on raster image are visible, these commands do not save any changes to the actual raster file.
It is strongly recommended to save raster image with EasyDone command "Save" or "Save As" after each raster editing operation. If you forget to do so and saved drawing with only AutoCAD's saving command, all raster changes will be missed when you close this drawing or quit AutoCAD. When you open such drawing next time and apply some EasyDone command, the original raster image (before changes) will be automatically restored and reattached to the drawing.

Export
When you call this command, the following dialog box appears.
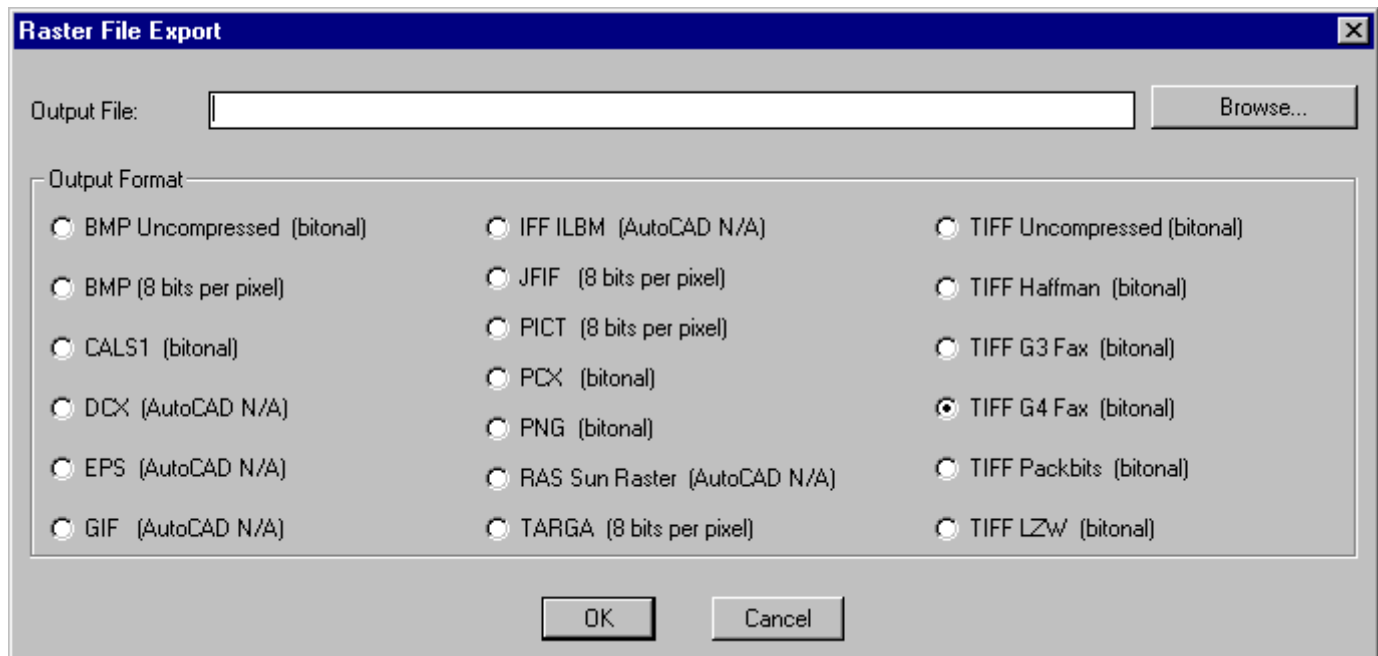
You may specify the desired path to output file, as well as required output raster format.
The difference between commands "Export" and "Save As" is that command "Export" does not attach saved raster image to the current drawing. It may be helpful, if you made some changes to raster image but do not want to save current DWG with this changed image. Command "Export" allows also saving of image in some formats not supported by AutoCAD, but which may be necessary for your other needs.

Select
This command may be used for two purposes.

1) If the only raster image is attached to the current drawing, you may use this command to change raster color to gray. It is recommended because the default color of attached raster images is BYLAYER and it varies depending on color of AutoCAD's working window (for instance, if window has black color, the color of raster image is white). White or black color of raster is not convenient to see recognized vectors or text above the raster background, but on a gray raster all recognized objects are visible much better.

2) If two or more raster images are currently attached to the drawing, you may use command "Select" to select the particular image, with which EasyDone must work now. When this work (for instance recognition or raster editing) is done, you may select another image for another work (note that command "Unselect" is not required to make a new selection).
Command "Select" changes raster color of the selected image to gray, as in the first case.

Unselect
This command restores color of raster image to the original. Note that it restores color only of the last selected image: if two or more images are attached to the drawing and you have selected more than one of them during current session, the color of the last selected image only may be restored.

Brightness
This command allows changing of brightness of raster gray color, applied by command "Select".
The following message appears in the command line, when you call this command:

Select brightness of gray raster: Esc/Low/High/<Low>:

or

Select brightness of gray raster: Esc/High/Low/<High>:

The selection **<Low>** or **<High>** indicates which setting is currently in use (so, if you press Enter or right mouse button then nothing will change). If you want to change selection, type "**h**" or "**l**" accordingly and then press Enter or right mouse button.
After that you need to call command "Select" to apply actual change of raster color.

Raster Operations: Menu "Raster Edit / Erase"

The commands of this group allow removing of areas, which should not be recognized, or removing of some unwanted raster objects. These commands include the following modifications.

Rectangle
Remove raster in the selected rectangular area. User defines this area by the rectangular frame around it.

Polygon
Remove raster in the selected polygonal area. User defines this area by picking vertices of polygon around it.

Circular Area
Remove raster in the selected circular area. User defines this area by picking the first and the second points on the diameter of the circle around it.

Line
Remove raster in the line-shaped area. User defines this area by picking the first and the second points of the line. To change width of the line, use "+" and "-" buttons.

Polyline
Remove raster in the polyline-shaped area. User defines this area by picking vertices of the polyline. To change width of the polyline, use "+" and "-" buttons.

Circle
Remove raster in the circle-shaped area. User defines this area by picking the first and the second points on the diameter of the circle. To change width of the line, use "+" and "-" buttons.

Spot
This command allows removing of raster in a square spot of the defined size. To perform operation user needs to point the necessary place on raster and press left mouse button (each erasing spot requires another pressing of left mouse button). To change size of the spot, use "+" and "-" buttons.

Raster Operations: Menu "Raster Edit / Draw"

The commands of this group allow drawing of some new objects on raster. These commands include the following modifications.

Rectangle
Fill raster in the selected rectangular area. User defines this area by the rectangular frame.

Polygon
Fill raster in the selected polygonal area. User defines this area by picking vertices of polygon around it.

Circular Area
Fill raster in the selected circular area. User defines this area by picking the first and the second points on the diameter of the circle around it.

Line
Fill raster in the line-shaped area. User defines this area by picking the first and the second points of the line. To change width of the line, use "+" and "-" buttons.

Polyline
Fill raster in the polyline-shaped area. User defines this area by picking vertices of the polyline.
To change width of the polyline, use "+" and "-" buttons.

Circle
Fill raster in the circle-shaped area. User defines this area by picking the first and the second points on the diameter of the circle. To change width of the line, use "+" and "-" buttons.

Spot
This command allows drawing of a square raster spot with defined size. To perform operation user needs to point the necessary place on raster and press left mouse button (each spot requires another pressing of left mouse button). To change size of the spot, use "+" and "-" buttons.

Raster Operations: Menu "Raster Edit / Modify"

Crop
Allows to crop rectangular or polygonal raster area.

Copy
Allows to copy rectangular or polygonal raster area to a new location within current image.

Move
Allows moving of rectangular or polygonal raster area to new a location current image.

Rotate90
Allows rotating of raster image by 90, 180 or 270 degrees.

Rotate
Allows rotating of raster image by any degree. User may define the rotation angle by typing its degree value or by picking the first and the second points of the vector, which defines the desired angle.

Deskew
This command may be used to align orientation of raster image relatively to the direction of some selected raster line (for instance to make it strictly orthogonal). To define the angle of alignment, user needs to pick the first and the second points of the current direction of the selected line and then to pick the first and the second points of the desired new direction of this line, or to type angle value from keyboard.

Remove Speckles
Provides automatic removing of small background noise. User defines the maximum pixel width and height of speckles, which should be removed, and selects the rectangular or polygonal area (or the whole image) where this operation should be applied. See illustration at the end of topic.

Fill Holes
Provides automatic filling of meaningless holes inside raster lines. User defines the maximum pixel width and height of holes, which should be filled, and selects the rectangular or polygonal area (or the whole image) where this operation should be applied. See illustration at the end of topic.

Smooth Edges
Allows to smooth edges of raster lines. User defines the rectangular or polygonal area (or the whole image) where this operation should be applied. See illustration at the end of topic.

Invert
This command may be used to invert colors of raster lines and background to the opposite (white pixels become black and vice versa).

Mirror
This command may be used get the mirrored raster image relatively vertical or horizontal axes.

Commands "Fill Holes", "Remove Speckles" and "Smooth Edges" may seriously enhance quality of raster image, which was originally too poor because of poor quality of the source paper document. See the following illustration of how these commands may improve raster image.

Figure 1. The illustration of the original raster image

Figure 2. The illustration of results of operation "Remove Speckles"

Figure 3. The illustration of results of operation "Fill Holes"

Figure 4. The illustration of results of operation "Smooth Edges"

This enhanced raster image becomes available for reliable recognition.

Table Converter: General Information

Table Converter is designed to convert tables realized by traditional AutoCAD objects (LINE, POLYLINE, TEXT, MTEXT, etc.) into Excel format and/or new format ACAD_TABLE, supported for the first time in AutoCAD 2005. Note that Excel conversion allows placing in the same output document a number of tables from the current drawing and even from different drawings.

1. Control of the process in AutoCAD+Excel conversion mode

This mode is invoked if parameter "Excel Conversion Settings/Disable AutoCAD Table conversion (Excel only)" is turned off. In this mode you may convert selected source table(s) in AutoCAD table format ACAD_TABLE and optionally in Excel format. When program is started in this mode, the following message appears in the command line:

[TABLE CONVERTER]Select table/F4=auth&stat/F7=zoom/F8=parameters/F9=help/Esc=exit

Info: Option **F4=auth&stat** may be used for product authorization and getting information about the number of remaining trial starts. If product is authorized, messages will appear without this option, so this case will be used further in this document for illustration of command line messages as the basic variant.

After you select the source table, the following message appears in the command line:

[TABLE CONVERTER]Press Enter or right mouse button to confirm selection or Esc

To start conversion in the selected area just press Enter or right mouse button, to cancel conversion press Esc button. When conversion is done, program shows resulting table converted into ACAD_TABLE format and displays the following message in the command line:

[TABLE CONVERTER]Check result (table): F4=table/F5=origins/F6=table+origins/F7=zoom/F8=Excel/Esc=return

If you would like to compare result of conversion with the source objects (origins), it is possible to do by pressing F6. In this case program will show converted table and source objects both, and will display the following message in the command line:

[TABLE CONVERTER]Check result (table+origins): F4=table/F5=origins/F6=table+origins/F7=zoom/F8=Excel/Esc=return

If you would like to discard result of table conversion and to return to the source objects, it is necessary to press F5. In this case program will delete converted table, restore source objects and display the following message in the command line:

[TABLE CONVERTER]Check result (origins): F4=table/F5=origins/F6=table+origins/F7=zoom/F8=Excel/Esc=return

If you would like to convert the same table in Excel format, it is possible to do by pressing F8 (option **F8=Excel),** independently on which result of conversion into ACAD_TABLE format is currently enabled: table, origins or table+origins. After that this table will be added to the temporary working file with information for Excel export, and the following message will be displayed.
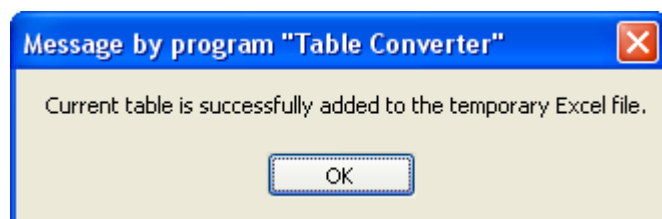
To return from "Check result" step press Esc. DWG will contain that result, which is currently enabled, that is: converted table and source objects, if F6 was pressed, source objects only, if F5 was pressed, or converted table and removed source objects, if F4 was pressed.

After that the following message will appear in the command line:

[TABLE CONVERTER]Select table/F5=saveExcel/F6=clearExcel/F7=zoom/F8=parameters/F9=help/Esc=exit

Note that tables converted to Excel are accumulated in the temporary working file until F6 (option **F6=clearExcel**) is pressed or Table Converter is closed (if parameter "Enable multi-DWG conversion

mode" is turned on, then tables will be accumulated even after Table Converter is closed, see below). To zero this process and to start accumulate tables for Excel from the very beginning, press F6.

If you want to view and/or to save Excel file with currently accumulated tables, press F5 (option **F5=saveExcel**). After that Table Converter will open output document directly in Excel program, where you may save it or just view and close Excel without saving (if you want to save it later). Note that Table Converter does not provide its own means for saving of Excel output documents, so saving is realized only through Excel program and is under complete control of user.

To continue conversion process with another table(s), just select another table. If you need to move to some different place on the drawing, you may press F7 for calling of AutoCAD command "Real Zoom" or use scroll-bars and/or mouse wheel (spinning of mouse wheel provides zooming capabilities, pressing mouse wheel and moving mouse provides panning).

Info: Option **F7=zoom** is provided specially for AutoCAD "Real Zoom" command because it is not possible to call AutoCAD commands directly from Table Converter while program is running. Real Zoom command allows to zoom in and to zoom out current view, to move it, to select viewing window, etc. To switch between viewing modes within this command, press right mouse button and select necessary option from the pull-down menu. To quit the command and to return to Table Converter mode, select from menu exit option or press Esc.

To finish work with Table Converter and to exit it, press Esc button.

Info: It is possible to use corresponding numeric keys instead of functional keys in all modes within session with Table Converter (for instance "5" instead of "F5", etc.).

2. Control of the process in "Excel only" conversion mode
This mode is invoked if parameter "Excel Conversion Settings/Disable AutoCAD Table conversion (Excel only)" is turned on. In this mode you may convert selected source table(s) in Excel format without simultaneous conversion in ACAD_TABLE format and, correspondingly, without changing of DWG file. Control of Table Converter is similar to the described above. The difference is that "Check result" step is absent and all selected tables are converted in Excel and accumulated in the temporary working file without this step.

3. Using multi-DWG conversion
This option is invoked if parameter "Enable multi-DWG conversion mode" is turned on, and may be used with AutoCAD+Excel and "Excel only" modes both. With this option you may continue accumulation of tables for Excel in the temporary working file even after you exit Table Converter and then start it again. It allows to open different DWG files and to convert desired tables from each DWG, obtaining at the same time all results in the same output Excel document.

Conversion process may be adjusted by parameters of Table Converter. For instance, you may set up for ACAD_TABLE conversion specific layer for converted table, different colors and lineweights for table border and grid lines, etc. However, it is not necessary to define any parameters for successful conversion because if some (or all) of parameters are not specified, program will use default values. Note that for conversion in ACAD_TABLE format it is not necessary to define any parameters for text objects (for instance color, font, oblique angle, etc.) because program will use corresponding values of the source text objects.

Conversion in ACAD_TABLE format has also some restrictions, related mostly to restrictions of the AutoCAD table format itself. For instance, it is not possible to combine BLOCK REFERENCE object with some other objects in one and the same cell. Text in the table must be orthogonal (i.e. with 0, 90, 180, 270 rotation angle). Text objects in the same cell cannot have different text styles, etc. However, you should not concern about any of such restrictions because program checks it automatically. If Table Converter finds some discrepancy between requirements of the AutoCAD table format and the source table configuration or content, it will notify you and cancel conversion process.

Table Converter: Select Conversion Area

To start conversion process it is necessary first to select the area in the drawing, which contains the source table. The selection is realized by clicking of two diagonal points outside of the source table (like any other area selection in AutoCAD), and the order in which these points are clicked does not matter.

There are some illustrations below how to select conversion area.

If the source table is standalone and does not touch any other graphics objects, the selection is the most simple.

_____
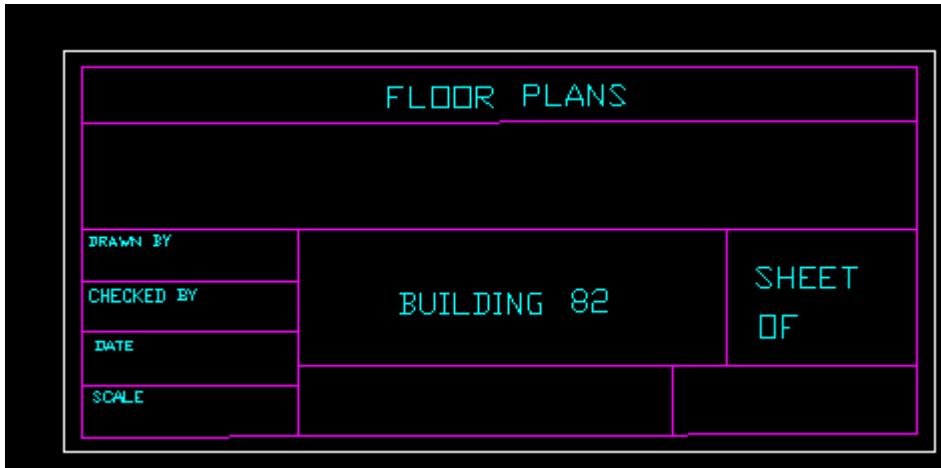
If the source table is located, for instance, in the corner of the drawing and drawing overall frame partially coincides with border of the table, the selection should be the same as when the table is standalone.
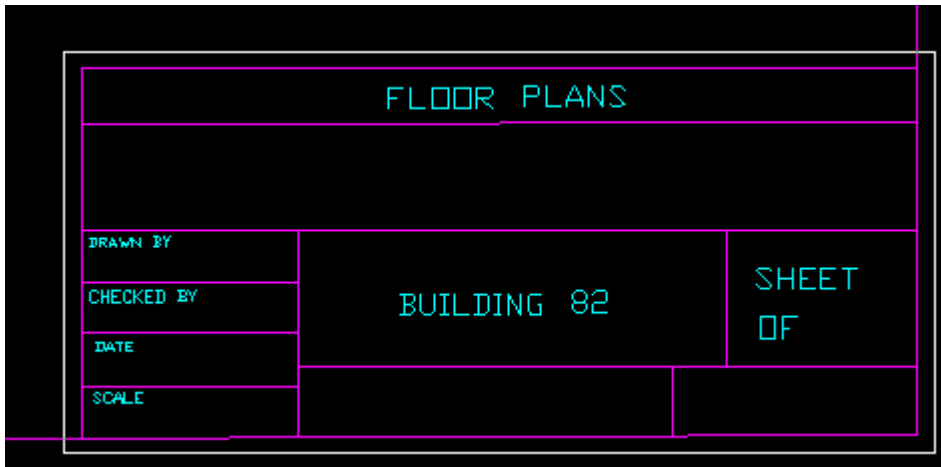
If table has two or more parts with different height (width), each part should be converted separately, as independent table. In this case the order, in which to convert parts, may be important or not. If there is some graphics line between converting parts, which continues outside of the largest part (for example it may be drawing overall frame), then the order of converting different parts does not matter because this graphics line will remain after conversion in any case.
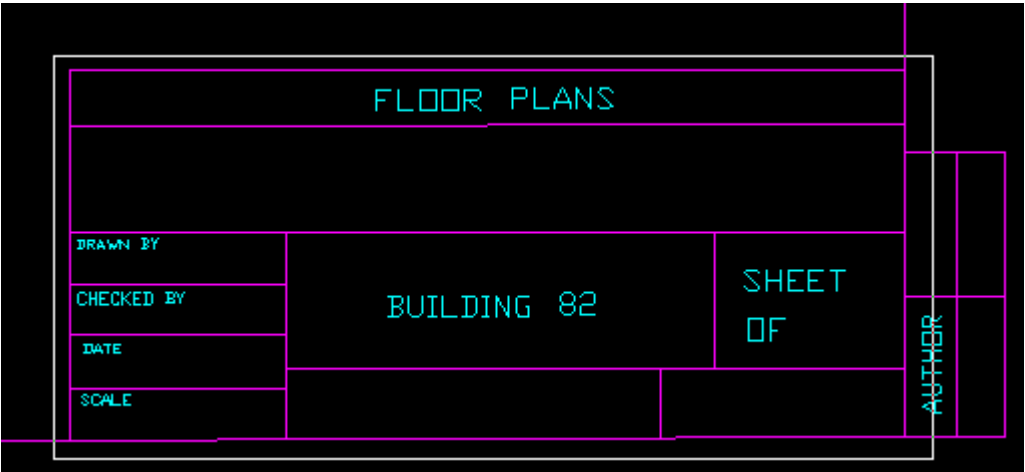
We may convert first the large part and then the small part.

_____
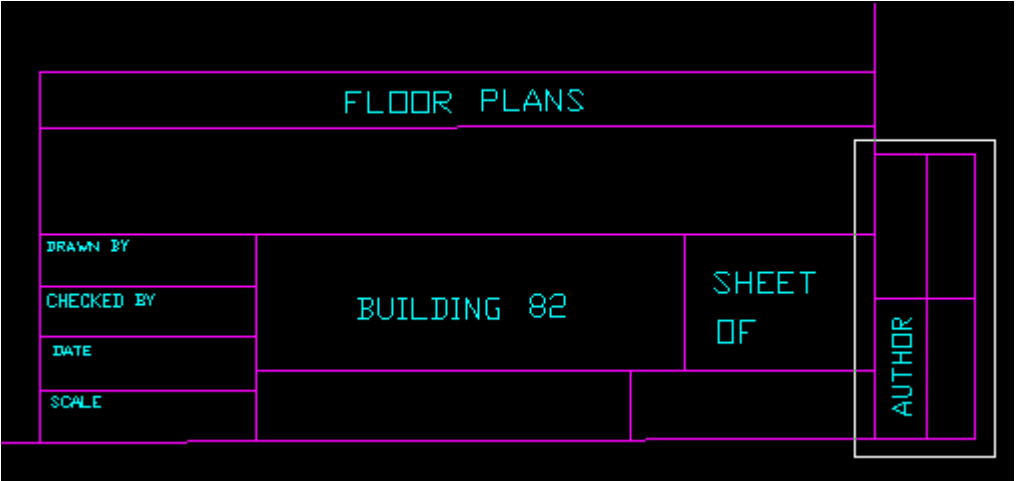
If smaller part merges directly with bigger part, it is important to convert first the smaller part. If we convert first the bigger part, it will result in removing of the common line between parts (because it is the border of the bigger part), so the smaller part will lose one of its borders.
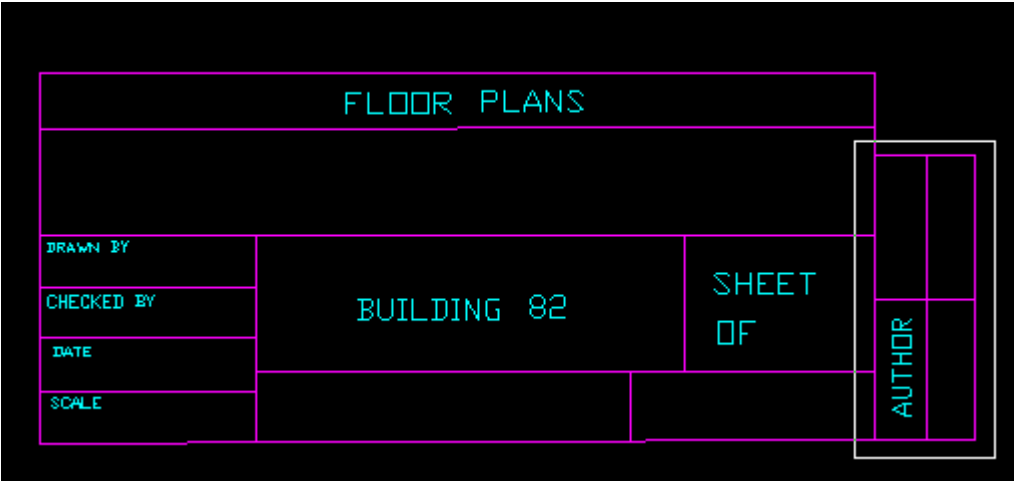
Note: This restriction is applied only for conversion in AutoCAD+Excel mode and is not related to conversion in "Excel only" mode.

Table Converter: Table Converter Parameters


Selection Type
This parameter allows switching between three different selection methods.

Rectangle is the usual method of area selection in AutoCAD. After user clicks the first point, the rectangle frame drags along with cursor movement that makes the selection process easier and more visible. This method may be recommended if the conversion area is located completely in the current view.

Rubber-band is selection method similar to selection by rectangle, but in this case the "rubber-band" is dragging with cursor instead of rectangle frame.

Two points is special selection method, which allows selecting conversion area by clicking of two "independent" points. In this case nothing drags with cursor after the first point is clicked. This method may be more convenient, for instance, when some outer graphics lines are located too close to the table

border. In this case user may change scale of the current view to see the table corner with higher resolution, that makes easier clicking of the first point (see illustration below).
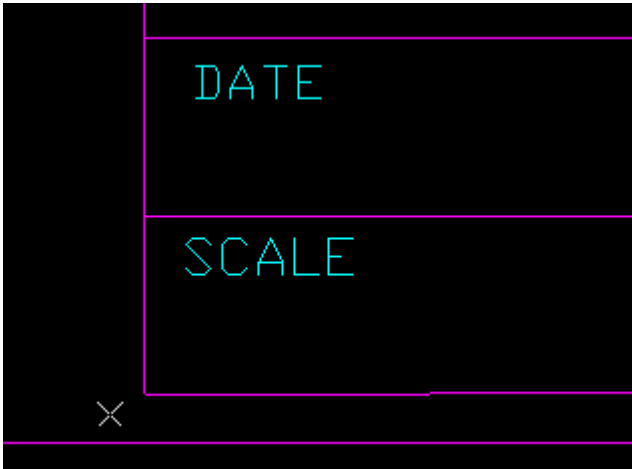
Then user should move to the diagonal corner for clicking of the second point. It is possible to do within "Two points" mode by the build-in zoom function (press F7), which allows using of zoom and pan commands directly inside Table Converter.
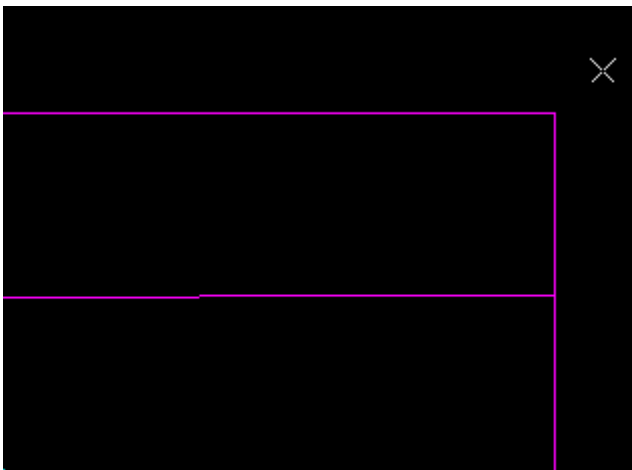
After the second point is clicked, program automatically changes the current view to display the whole conversion area.

AutoCAD Table Layer
These parameters specify the layer on which the converted table will be placed (not used with "Excel only" conversion mode).

Select layer from the drawing
This option allows selecting of the layer, on which the converted table will be placed, from layers existing in the current drawing.

Create new layer
This option allows to create new layer, on which the converted table will be placed, and to define color setting for this new layer.

AutoCAD Table Conversion Settings
These parameters define specifics of conversion in ACAD_TABLE format.

Table border color and lineweight
These options allow setting up color and lineweight for border of the converted table.

Horizontal grid color and lineweight

These options allow setting up color and lineweight for horizontal lines of the internal table grid. These parameters may have different values from color and lineweight of table border or vertical grid lines.

Vertical grid color and lineweight

These options allow setting up color and lineweight for vertical lines of the internal table grid. These parameters may have different values from color and lineweight of table border or horizontal grid lines.

Display lineweight

This parameter enables or disables displaying of object lineweights on the screen.

Lineweight units (mm or inches)

This parameter defines whether lineweight values are listed in millimeters or inches.

Alignment of text in cells

This parameter defines how text is aligned in cells of converted table.

"Positioned" option may be used to align text in cells according to positions of the source text objects. Note that converted text cannot coincide with original objects exactly because AutoCAD does not provide any possibility to locate text in cells at an exact position. However Table Converter will try to locate text in converted cells as closer to the original positions as possible.

"Centered" option allows aligning text in cells centered.

If converted text does not fit in a cell

This parameter defines what program should do if converted text cannot fit in some cell. Such situation may occur, for instance, if lines of the source text are written too close to each other in some cell. AutoCAD table format does not support line spacing for text in cells, so if lines of source text are written too close, they may not fit in the cell after conversion.

"Decrease text size" option allows program to decrease height and/or width of the converted text until text can fit in the cell and to keep original size of the whole table because of that.

"Increase table size" option requires program to keep the original text size, and allows to increase size of the whole table.

If program cannot identify some grid line

This parameter defines what program should do if some grid line is not properly connected to another grid lines or table border. Such situation may occur, for instance, if there is big gap between end of grid line and another line, to which this line should be connected.

"Notify and stop" option requires program to notify user about the situation and to cancel execution.

"Delete this line" option allows program to delete the problem line and continue execution. However, it is not recommended to use this option unless the source table really contains some graphics lines, which should be deleted, because some actual grid lines may be removed, if this option is used.

If cell contains diagonal grid line

This parameter defines what program should do if some cell contains diagonal grid line.

"Convert to horizontal" option requires program to convert this diagonal line to horizontal line and to split the cell into two cells merged vertically.

"Convert to vertical" option requires program to convert this diagonal line to vertical line and to split the cell into two cells merged horizontally.

"Delete" option requires program to delete the diagonal grid line.

Excel Conversion Settings
This group of parameters defines how to convert tables in Excel format.

Disable AutoCAD Table conversion (Excel only)
If this parameter is turned on, the conversion in ACAD_TABLE format will be disabled (see topic "*General Information*" for details).

Enable multi-DWG conversion mode
This parameter allows collecting of converted tables from different DWG files in one and the same output Excel document (see topic "*General Information*" for details).
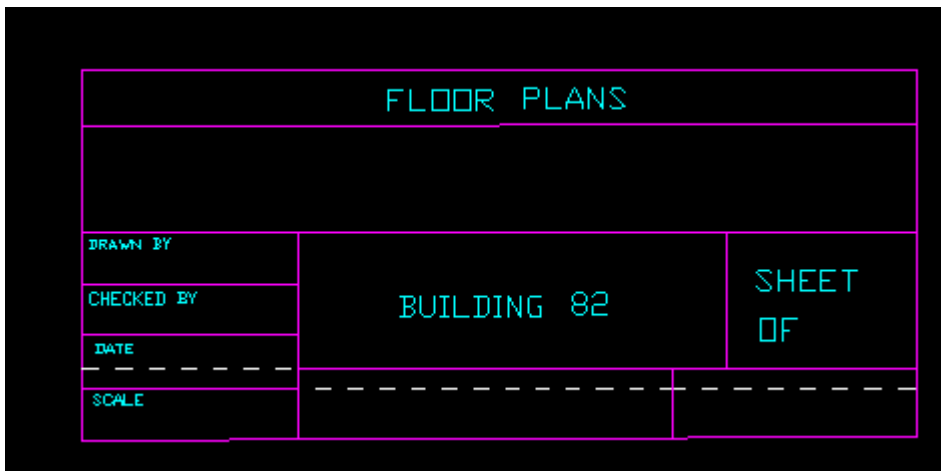
Remove empty columns
If this parameter is turned on, program will automatically remove empty columns in each single table. If empty columns in some table are meaningful, this parameter should be turned off.

Remove empty rows
If this parameter is turned on, program will automatically remove empty rows in each single table. If empty rows in some table are meaningful, this parameter should be turned off. This parameter does not affect blank rows inserted by program between single tables according to the parameter "Number of blank rows to insert between single tables".

Note 1: If parameter "Remove empty columns" and/or "Remove empty rows" is turned on, it may slow down the processing of the output document in Excel because program will remove empty columns/rows by moving other columns/rows onto empty places, like it is implemented in "garbage collection" algorithm. This process will be also visible on the screen in real time mode.
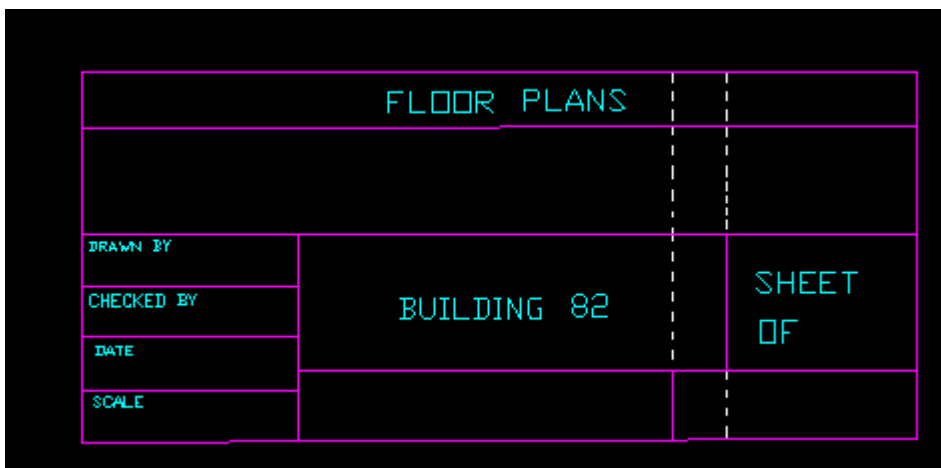Note 2: Note that empty columns or rows sometimes may appear even if they do not look like "visible" in the source table. It may happen, for instance, if this table has merged cells of some specific configuration (see illustration below).

The reason of appearing of "invisible" empty row

The reason of appearing of "invisible" empty column

AutoFit column width
This parameter enables Excel "AutoFit" formatting option for table columns.

AutoFit row height
This parameter enables Excel "AutoFit" formatting option for table rows.

Number of blank rows to insert between single tables
This parameter allows inserting of blank rows between single tables to separate them from each other for better reading. If such separation is not required, turn this parameter to zero.

Path to output folder
This parameter defines location of the folder for saving of output Excel documents.

Browse
This button calls standard dialog box, which you may use to find output folder. If folder does not exist, you may create it with this standard dialog box.

Default Setting
This button discards all current parameter values and sets up all parameters to the default values.

Additional Operations: Menu "Measure Size"

Command "Measure Size" may be invoked by menu "EasyDone/Measure Size", by EasyDone toolbar button marked "MS", or by pressing button "Measure" in some EasyDone dialog box.
This command allows to measure height, width (in pixels and World Coordinates), and angle of any object, which you outlined by the measuring frame. You may use this feature to measure all graphical and text objects, which size or angle is required for setting up corresponding values of different parameters. The result of measuring will appear in the command line and at the bottom of the dialog box (if you call this command from a dialog box).

The following illustration shows how to measure height, width and angle of a word on raster.
After command starts, it prompts to pick the first point of the measuring frame. Click this point by left mouse button at the left or right, lower or upper corner of the word on raster (it is lower left corner in the following sample).

Then drag cursor to the opposite corner on the bottom, top, left, or right line (bottom in this sample) and press left mouse button.
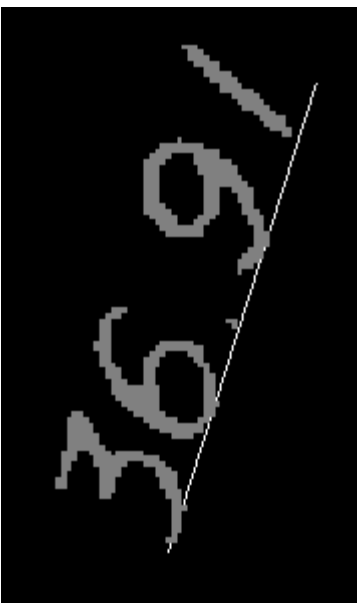
Then drag cursor in the perpendicular direction (to the top of the word in our case) and then press left mouse button again.
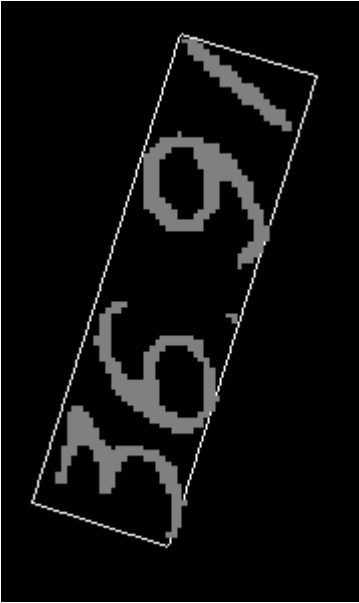
You will get measuring information in the command line.
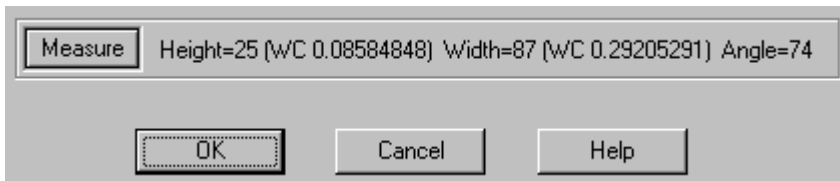To complete measuring operation, press Enter or right mouse button.

If you called command "Measure Size" from a dialog box, you will see the following result:

Additional Operations: Menu "Reset Warnings"

When you work with EasyDone, you will receive some "turn-off" messages with instruction or advise what to do in some situation. You may turn each of such messages OFF, so this message will not appear any more.

If you forgot what was written in some message, you may turn all messages ON with command "Reset Warnings", and they will be available again.

Note that this command enables ALL messages, so you will need to turn OFF each of unwanted messages again.

Additional Operations: Menu "Convert LW to 3D polylines"

EasyDone provides graphics recognition and editing operations with "light weight" polylines (AutoCAD entity LWPOLYLINE). However, some users may need 3D format of polylines.
Command "Convert LW to 3D polylines" allows converting of LWPOLYLINE to 3D format on the whole drawing at once.

TIP: Because 3D polylines cannot be modified by Recognition Result Editor, you should complete all required editing operations before converting. Note that polylines in 3D format do not have parameter "width" so all width related information would be missed after converting.

Converting to 3D format requires that "light weight" polylines must not contain any arc-segments (vertices with bulge attribute), because such attribute is absent in 3D format. If you need to use arcs, you may turn ON parameter "Arc_Entity" in Arc Recognition Parameters dialog box (see the topic "Arc Recognition Parameters"). In this case arcs and circles will be recognized as ARC and CIRCLE AutoCAD entities that is compatible with 3D polyline format.

Additional Operations: Menu "Create DIAGNOSE file"

If EasyDone works improperly with some image, it is recommended to create DIAGNOSE file and to send it to RECSOFT Inc., otherwise technical support may be not possible. This file includes all EasyDone working files, related to the current parameter setting, as well as current drawing file and the attached raster image. It is important to create this file immediately after wrong situation has occurred, otherwise parameter setting may be changed and it would be impossible to recreate exactly the same situation for technical support.

After you call command "Create DIAGNOSE file" the file DIAGNOSE.BIN will be created in the EasyDone working directory. Send this file and your description of the wrong situation to:
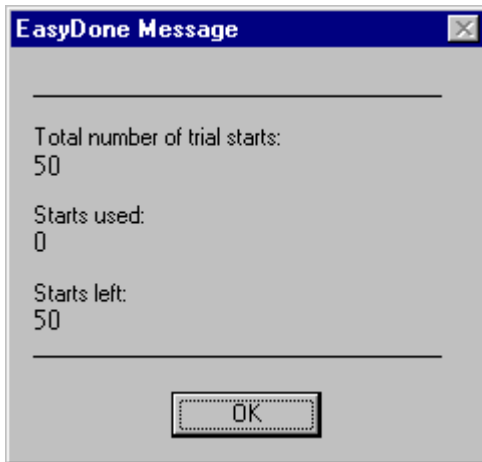
support@recsoft.com

Note: If the drawing or attached raster image contains any kind of confidential or classified information, you MUST NOT create file DIAGNOSE.BIN and send it to RECSOFT Inc.

Display Statistics of Trial Starts

When you work in trial mode it is recommended to check time-to-time how many free starts left, especially if you tried Batch Recognition, in which each recognized image reduces count of remaining free starts. To display statistics of trial starts you may type in command line command:

123

The following message will appear after that.

```
EasyDone Message                    [X]
_____

Total number of trial starts:
50

Starts used:
0

Starts left:
50
_____


          [       OK       ]
```

Because this information is related only to trial mode, i.e. is used just temporarily, this call is not included in the EasyDone menu and may be invoked only by the above command.

Authorization (switch to permanent working mode)

To switch EasyDone from trial to permanent working mode it is necessary to acquire special Authorization Password, which may be purchased on-line through our site:
**www.recsoft.com**

To purchase the authorization password you will need EasyDone Serial Number and Verification Code, which you may find in the panel "About". The Serial Number will be used for automatic generating of the authorization password for your copy of the product. The Verification Code will be used to verify that the provided Serial Number is correct and is typed without mistakes.
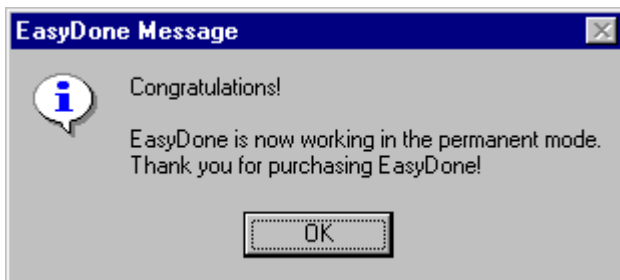
After the authorization password is acquired, it is necessary to start EasyDone and to type in the command line the following command:

_EDAUTH

After that the following panel will appear.
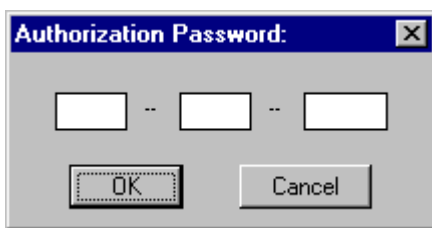
When you type in this panel the acquired Authorization Password and press button OK, the following message will appear.

After that EasyDone will work in the permanent working mode without any further authorizations and without any hardware lock.

Note that typing in command line **123** or **_EDAUTH** will produce no action after product is authorized because these commands are used only in trial mode.

If you want to re-install authorized product on another computer or with different version of Windows, you should obtain new authorization password, which you may receive for free. Our company provides special service for that (see the topic "Instructions for Re-installation" for details).

Instructions for Re-installation

If EasyDone is not authorized yet, i.e. is working in trial mode, then re-installation may be done usual way.

If you authorized EasyDone and want to re-install it on the <u>same computer</u> and with the <u>same version of Windows</u>, re-installation will not have any peculiarity as well and may be done usual way (after uninstallation and new installation your copy will remain authorized). You may perform new install in the same or in some different working directory - it will not affect authorization information because this information is located in the Windows Registry.

If you want to use authorized copy of EasyDone on another computer then, according to License Agreement, you should remove the product from the previous computer. After removing of the previous version of EasyDone, you may obtain new Authorization Password for FREE (the same is for the case if you need to use authorized copy with different version of Windows).

Note: The term "different version of Windows" does not necessarily mean switching to different platform (for instance from Windows 98 to Windows XP). If you install on the same computer the same version of Windows that you had earlier, but **from the beginning**, it will result in loss of the previous Registry, which contains EasyDone authorization information, so you will need free new Authorization Password in this case as well.

To obtain free new Authorization Password it is necessary first to remove from computer system components of your current authorized version of EasyDone. To do so you need to start EasyDone and to type in the command line the following command:

_EDUNCOMP

After that the following message will appear.
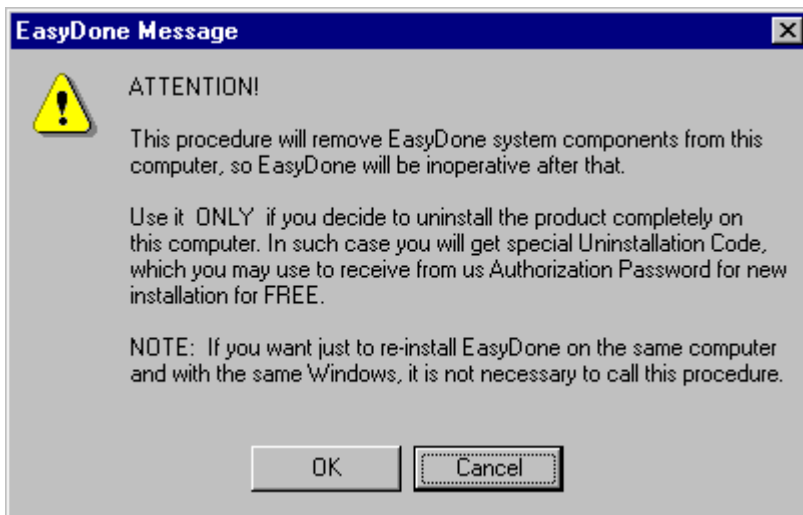
After you press button OK, the following confirmation message will appear.
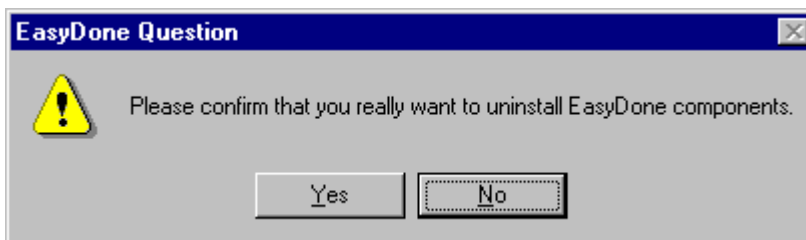
If you press button "Yes", the product system components will be removed and the following message will appear.

EasyDone Message ☒

ⓘ Old Serial Number: xxx - xxx - xxxx

Uninstallation Code: xxx - xxx - xxxx

Please retain this information! For your convenience it is automatically saved in the file "Easydone.inf" located in the root of the harddrive where EasyDone was installed.

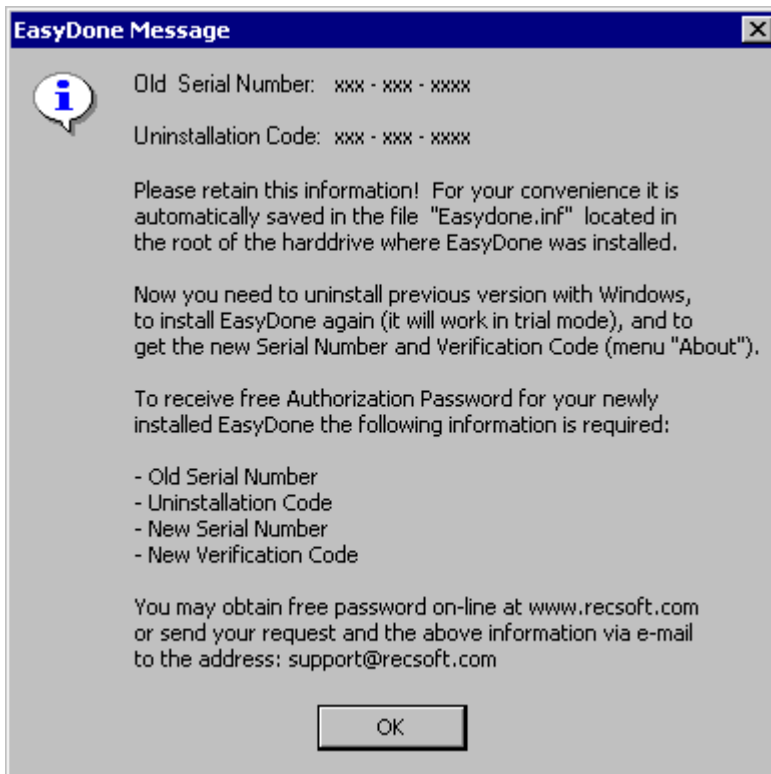Now you need to uninstall previous version with Windows, to install EasyDone again (it will work in trial mode), and to get the new Serial Number and Verification Code (menu "About").

To receive free Authorization Password for your newly installed EasyDone the following information is required:

- Old Serial Number
- Uninstallation Code
- New Serial Number
- New Verification Code

You may obtain free password on-line at www.recsoft.com or send your request and the above information via e-mail to the address: support@recsoft.com

[OK]

Note: There are symbols "x" instead of actual numbers in this sample.

It is recommended to write Old Serial Number and Uninstallation Code on paper. For your convenience this information is also automatically saved in the file "Easydone.inf" in the root of the harddrive, where EasyDone working directory is located. After you wrote this data on paper, it is necessary to press button OK, to quit AutoCAD and to uninstall EasyDone under Windows (you may use for that, for instance, command "Add/Remove Programs" in Windows "Control Panel"). Then you may install EasyDone on the new computer or with new version of Windows (EasyDone will work in trial mode). When new copy of EasyDone is installed, you need to open EasyDone panel "About" and write on paper new Serial Number and Verification Code.

After all necessary information is collected, you may visit page "Re-installation password" on our site and obtain new Authorization Password for your new copy of EasyDone for free. Note that you may receive free authorization passwords for re-installation as many times as you need.

Creation of Additional Copies

Sometimes it may be convenient to keep on the same computer more than one copy of the product. For instance, one copy may be used for batch recognition and another for processing of single drawings, or one copy may be used as working and another as a test-copy. Unfortunately it is not possible to use installation procedure for creation of additional copies because Setup first removes previous copy of the product and installs new copy only after that. However, it is easy to create additional copies of EasyDone manually. To create additional copy of EasyDone you may do the following.

1) Copy EasyDone working directory and all its sub-directories into some new folder.

2) Copy EasyDone shortcut (click EasyDone shortcut by right mouse button and then select in the pull-down menu option **Copy**).

3) Insert copied shortcut for instance on Desktop (click any empty place on Desktop by left mouse button, then press right mouse button and select in the pull-down menu option **Paste Shortcut**).

4) Click copied shortcut by right mouse button and select in the pull-down menu option **Properties**).

5) Place in the field "Start in" path to the folder, in which you have copied EasyDone working directory on step 1 (see the field highlighted by red oval on the illustration below).
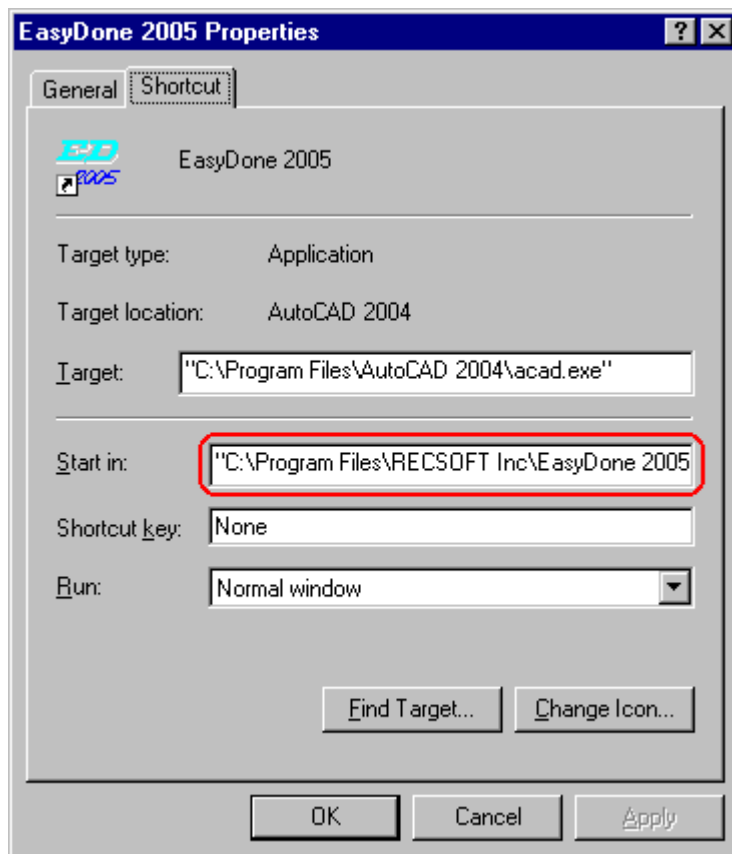
6) Press OK button to close **Properties** panel.

Now you may use this new shortcut to call copy of EasyDone, and the old shortcut to call original version of EasyDone. If EasyDone was originally authorized, then the new copy will be authorized too. Each copy may have its own parameter setting, which does not interfere with parameters of another copy.

Tutorial Samples

It is recommended to start learning of how to work with EasyDone, using "Demo Tutorial" demonstration system provided on EasyDone CD. It is the most complete source of learning assistance. We decided do not include information from "Demo Tutorial" in this document because, otherwise, it will be too big and inconvenient for reading or printing. Note that in addition to "Demo Tutorial" you may learn how to use particular editing operations with run-time demos, which you may call from help topics, associated with each editing operation.

There is the list of tutorial samples, supplied with EasyDone, and the brief description for which purposes they are included in EasyDone installation.

Graphics-Corners.tif
Use this sample for getting practice with editing operation "Amend Corners". It is recommended to pass through this operation first with "Default" parameter setting, and then with "Extremal" setting. It will help to understand what effect parameters have on situations that will be shown by program within this operation.

Graphics-Outline.tif
This sample is just the illustration of how raster outlining works.

Graphics-Solids.pcx
This sample illustrates recognition of solids. You may use it for training with solid editing operations.

Sample.tif
Recognition and editing of this sample is considered in all details in "Demo Tutorial". This image includes the most complete set of entities and may be used for training with all editing operations.

Text-Angle.tif
This sample illustrates text recognition with oblique angle. You may use it to learn how different oblique angles affect the obtained result.

Text-Dictionary.tif
This sample allows comparing of result of text recognition with and without text dictionary, which is prepared specially for this sample (file "Text-Dictionary.trd" in folder <Dictionary Files> in EasyDone working directory). It is recommended to recognize this image first with option "Dictionary" (in Control Options dialog box) turned ON, and then with turned OFF. You will be able to compare results and learn how using of dictionary may improve reliability of text recognition.

TIP: Each tutorial sample has associated parameter set with the same name. You should open the corresponding parameter set before you start recognition of each tutorial image. You may also compare parameter settings for different tutorial samples and learn how to tune parameters for different types of drawings.

Technical Support

EasyDone has passed very thorough testing and quality assurance in all modes. However, some potential bugs may remain undetected, as in any software product. If you find such bug, we will try to fix it as soon as possible.

If some wrong situation occurs when you work with EasyDone dialog boxes, we will appreciate if you send description of all steps, which precede this wrong situation, so we could recreate it.

If a wrong situation occurs when you recognize or edit some raster image, we need also this image and EasyDone working files to recreate the same situation. In this connection we appreciate if you create file DIAGNOSE.BIN (see the topic "Create DIAGNOSE file") and send it via e-mail. Without this file we cannot repeat the wrong situation and cannot provide technical support. That is why we do not provide technical support via fax, regular mail or phone call.

Please send your request for support and associated information to the address:
support@recsoft.com