



# Converting Between BRL-CAD and Other Formats

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# Converters to/from ASCII

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## g2asc/asc2g

- Release 6.0 ASCII format is a Tcl script.
  - Database format is machine independent (for IEEE floating point architectures).
  - No need to convert to ASCII to move database to different machines.
- Old ASCII format still supported by asc2g (creates new database format).



# Converter to ASCII

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- `g2asc database.g database.asc`
  - works for all databases
- `g2asc < database.g > database.asc`
  - works only for pre-release 6.0 databases
- Pre-release 6.0 databases get converted to old ASCII format



# Converter from ASCII

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asc2g database.asc database.g

- If *database.asc* is new format, executed in a safe Tcl interpreter.
- If old format, each record is read and converted to new database format.
- Any *polysolids* get converted to BOT primitives.



# Upgrading to New Format

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`dbupgrade old_database.g new_database.g`

- Converts a pre-release 6.0 database to the current format.
- Any *polysolids* get converted to *BOT* primitives.



# CSG vs. BREP

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- Constructive Solid Geometry (CSG)
  - BRL-CAD
  - A part is a Boolean combination of primitive shapes .
- Boundary Representation (BREP)
  - Most commercial CAD systems
  - A part is a group of surfaces that define the boundary of the part .



# CSG vs. BREP (cont'd)

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- Converting CSG to BREP
  - Convert each primitive of a part to faceted BREP, then perform Boolean operations, output each part as a faceted BREP
- Converting BREP to CSG
  - Convert surfaces of each part to polygons, output each part as a faceted primitive



# Converters from BRL-CAD

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- g-acad
- g-tankill
- g-vrml
- g-wave
- g-x3d
- g-euclid\*

\*Not to native format



## g-acad

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- Converts BRL-CAD objects to faceted ACAD format.
  - ACAD is a CAD package developed by the Lockheed Fort Worth Company and distributed by the Electromagnetic Code Consortium (EMCC) as of 1995 (used by *xpatch*).
  - It is **not** *AutoCad*.



# g-acad (cont'd)

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g-acad [options] –o output\_file input.g object(s)

Options:

- i - output inches (default is millimeters)
- x lvl - set librt debug flag (see *raytrace.h* for definitions of DEBUG\_xxx)
- X lvl - set NMG library debug flag (see *nmg.h* for definitions of DEBUG\_xxx)
- e error\_file - send error messages to specified file (default is stderr)
- v - verbose, print out progress information



## g-acad (cont'd)

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- Options (cont'd):
  - D tolerance\_distance - set distance calculation tolerance (mm) (default is 0.005)
  - r rel\_tess\_tol - set relative tessellation tolerance (normally 0.0 to 1.0, default is 0.1)
  - a abs\_tess\_tol - set absolute tessellation tolerance to specified value (millimeters) (overrides -r)
  - n norm\_tess\_tol - set surface normal tessellation tolerance (angle in radians) (overrides -r)



# g-tankill

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Converts BRL-CAD objects to the British  
TANKILL format (Triangles)

g-tankill [options] input.g object(s)



# g-tankill (cont'd)

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## Options:

- s surroundings\_code - set surroundings code to use (default is 1000)
- i idents\_output\_file - output region idents to the specified file
- o output\_file\_name - set output file name (required)

Also accepts v, r, a, n, x, and X options



# g-vrml

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Converts BRL-CAD objects to VRML 2.0 format

`g-vrml [options] input.g object(s)`

Options:

- d tolerance\_distance - set distance calculation tolerance (mm) (default is 0.005)
- u units - set desired output units (default is "mm")
- o output\_file\_name - set output file name (default is stdout)

Also accepts v, r, a, n, x, and X options



# g-wave

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Converts BRL-CAD objects to Wavefront ".obj" format

g-wave [options] input.g object(s)

Options:

- m - include "usemtl" statements in the output (encoded aircode, los, and material number)
- i - set output units to inches (default is mm)
- D tolerance\_distance - set distance calculation tolerance (mm) (default is 0.005)
- u - include vertexuse normals in the output

Also accepts v, r, a, n, x, and X options



# g-x3d

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Converts BRL-CAD objects to X3D format

`g-x3d [options] input.g object(s)`

Options:

- d tolerance\_distance - set distance calculation tolerance (mm) (default is 0.005)
- u units - set output units (default is "mm")
- o output\_file\_name - set output file name (default is stdout)

Also accepts v, r, a, n, x, and X options



# g-euclid

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Converts BRL-CAD objects to a Euclid  
"decoded" faceted format  
(not to native Euclid)

`g-euclid [options] input.g object(s)`

Options:

- u units - set output units (default is "mm")
- o output\_file\_name - set output file name  
(default is stdout)

Also accepts v, r, a, n, x, and X options



# brcad/conv/g-xxx.c

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- Skeleton code for developing a converter from BRL-CAD to any faceted/triangular format
- May need updating to include new primitives (BOT, Extrusion, DSP)



# Converters to BRL-CAD

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cy-g

dxf-g

enf-g

euclid-g

fast4-g

patch-g

nastran-g

proe-g

Pro/Engineer 2001

stl-g

tankill-g

viewpoint-g

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## cy-g

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Converts Cyberware Digitizer Data (laser scan data) to a single BRL-CAD ARS primitive. The data must be in cylindrical scan format.

cy-g input\_file output\_file.g



## dx<sub>f</sub>-g

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Converts the AutoCad® DXF format to BRL-CAD. This was developed prior to AutoCad's use of ACIS® and does not support ACIS objects in the DXF file. Converts only *3DFACES* and *3DMESH* entities. Produces a single primitive.



# dxfg (cont'd)

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## dxfg [options]

Options:

- v - verbose, print out progress information
- p - produce BOT primitives (default is NMG primitives)
- d tolerance\_distance - set distance calculation tolerance (mm) (default is 0.005)
- i input\_file - set input file name (default is stdin)
- o output\_file\_name - set output file name (default is stdout)



## enf-g

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Converts an Elysium™ neutral faceted file to BRL-CAD. Each part is converted to a BRL-CAD region consisting of one BOT primitive. Ident numbers are incremented for each region. If a part name mapping file is provided, part names in the input file will be output using the corresponding names from the mapping file.



# enf-g (cont'd)

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enf-g [options] input\_file output.g

## Options:

- i starting\_ident - set the starting ident number (default is 1000)
- n part\_name\_mapping - set the mapping from input names to output names
- t tolerance\_distance - set distance calculation tolerance (mm) (default is 0.005)



# euclid-g

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Converts an ASCII Euclid[] "decoded" format file to BRL-CAD. Each part is converted to a BRL-CAD region consisting of a single primitive.

euclid-g [options]

Options:

- i input\_euclid\_db - set input file name (default is stdin)
- o output\_brlcad\_db - set output file name (default is stdout)
- d tolerance\_distance - set distance calculation tolerance (mm) (default is 0.005)
- p - produce BOT primitives (default is NMG )

Also accepts v, x, and X options



# fast4-g

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Converts FASTGEN4 to BRL-CAD.

FASTGEN4 is a faceted format used by the Navy and Air Force.

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fast4-g [options] fastgen4_input output.g
```

Options:

- d - print verbose debugging output
- q - print nothing except errors
- w - print warnings about creating default names
- o plot\_file\_name - create a Unix plot file of all CTRI and CQUAD elements processed



# fast4-g (cont'd)

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## Options (cont'd):

- c component\_list - process only the listed region ids, may be a list (3001,4082,5347) or a range (2314-3527)
- m muves\_file\_name - create a MUVES input file containing *CHGCOMP* and *CBACKING* elements
- b lvl - set libbu debug flag (see *bu.h* for definitions of BU\_DEBUG\_XXX)
- x lvl - set librt debug flag (see *raytrace.h* for definitions of DEBUG\_XXX)



# patch-g

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Converts pre-processed FASTGEN version 3 to  
BRL-CAD.

**patch-g [options] output.g**

Options:

- f fastgen.rp - specify pre-processed fastgen file (default stdin)
- a - process phantom armor
- n - process volume mode as plate mode
- u # - number of union operations per region (default 5)
- c \"x y z\" - center of object in inches (for some surface normal calculations)
- t title - optional title (default "Untitled MGED database")
- o object\_name - optional top-level name (default "all")



# patch-g (cont'd)

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## Options (cont'd):

- p - write volume and plate mode components as BOT's
- 6 - process plate mode triangles as ARB6 solids  
(overrides '-p' for triangles)
- i group.file - specify group labels source file
- m mat.file - specify materials information source file
- r - reverse normals for plate mode triangles
- d lvl - set debug level
- x lvl - set librt debug flag
- X lvl - set librt NMG debug flag
- T tolerance\_distance - set distance tolerance (inches)
- A parallel\_tolerance - set parallel tolerance (sine)



# nastran-g

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Converts NASTRAN files to BRL-CAD. Currently only converts "CBAR," "CROD," "CTRIA3," and "CQUAD4" elements.

**nastran-g [options]**

Options:

- i NASTRAN\_input\_file - set input NASTRAN file (default is stdin)
- o output.g - set output file name (default is "nastran.g")
- n - produce NMG primitives (default is BOT primitives)



# nastran-g (cont'd)

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## Options (cont'd):

- x lvl - set librt debug flag (see *raytrace.h* for definitions of DEBUG\_XXX)
- X lvl - set NMG library debug flag (see *nmg.h* for definitions of DEBUG\_XXX)
- t tolerance\_distance - set distance calculation tolerance (mm) (default is 0.005)
- m - set input units to millimeters (default is inches)



## proe-g

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Converts an ASCII ".brl" file to a BRL-CAD database. This is used with the original Pro/Engineer® to BRL-CAD converter for Pro/E releases prior to Pro/E 2001.

proe-g [options] proe\_file.brl output.g

Options:

- d - print additional debugging information.
- i starting\_ident - set the initial region ident number (default is 1000).
- I ident\_number - set the non-negative ident number that will be assigned to all regions (conflicts with -i).



# proe-g (cont'd)

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## Options (cont'd):

- -u reg\_exp - indicates that portions of object names that match the regular expression should be ignored.
- a - create BRL-CAD 'air' regions from everything in the model.
- r - the model should not be re-oriented or scaled
- S - the input file is raw STL (Stereo Lithography) format.
- t tolerance\_distance - set distance calculation tolerance (mm) (default is 0.005)
- x lvl - set librt debug flag (see "raytrace.h" for definitions of DEBUG\_xxx)



# Pro/Engineer 2001

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Uses "protk.dat" file to install BRL-CAD converter as part of Pro/Engineer. New menu item named "Proe-BRL" installed in Pro/E "File" menu. Use this menu item after loading a Pro/E model. A Pro/E dialog box will pop up where you can enter the output file name, starting ident number, and facetization control factors. Output is the new BRL-CAD ASCII format (a Tcl script). Use "asc2g" to convert this to a BRL-CAD database.



# stl-g

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Converts an ASCII stereo lithography file to a BRL-CAD database. Creates a single BRL-CAD region consisting of a single BOT primitive.

`stl-g [options] input_stl_file output.g`

Options:

- `c units_str` - specifies the units used in the STL file (default is "mm")
- `N object_name` - specifies a name to use for the output object.
- `d` - print additional debugging information.
- `I ident_number` - set the ident number that will be assigned to the region.



# stl-g (cont'd)

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## Options (cont'd):

- m material\_code - set the integer material code for the region (default is 1).
- u reg\_exp - indicates that portions of object name that match the regular expression 'reg\_exp' should be ignored.
- a - create a BRL-CAD 'air' region from everything in the model.
- t tolerance\_distance - set distance calculation tolerance (mm) (default is 0.005)
- x lvl - set librt debug flag (see *raytrace.h* for definitions of DEBUG\_xxx)



# tankill-g

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Convert British TANKILL format to BRL-CAD  
tankill-g [options]

Options:

- v - verbose, print out progress information
- p - produce BOT primitives (default is NMG primitives)
- k - keep components with id = 1001 (normally skipped)
- x lvl - set librt debug flag (see *raytrace.h* for definitions of DEBUG\_xxx)
- X lvl - set NMG library debug flag (see *nmg.h* for definitions of DEBUG\_xxx)



## viewpoint-g

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Convert the Viewpoint Datalabs coor/elem format to BRL-CAD format. Will assign vertex normals if they are present in the input files. Two files are expected, one containing vertex coordinates (and optional normals) and the second which lists the vertex numbers for each polygonal face.



# viewpoint-g (cont'd)

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## viewpoint-g [options]

Options:

- c coord\_file\_name - set the input vertex coordinates file name (required)
- e elements\_file\_name - set the input faces file name (required)
- o output\_file\_name - set the output BRL-CAD database name (default is "viewpoint.g")
- t tolerance\_distance - set distance calculation tolerance (mm) (default is 0.005)



# Future Converter Plans

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- Unigraphics to BRL-CAD converter currently under testing, should be in next release.
- CATIA to BRL-CAD converter planned for this FY.
- g-stl converter planned for this FY.
- Improvements to converters from commercial systems using construction history.