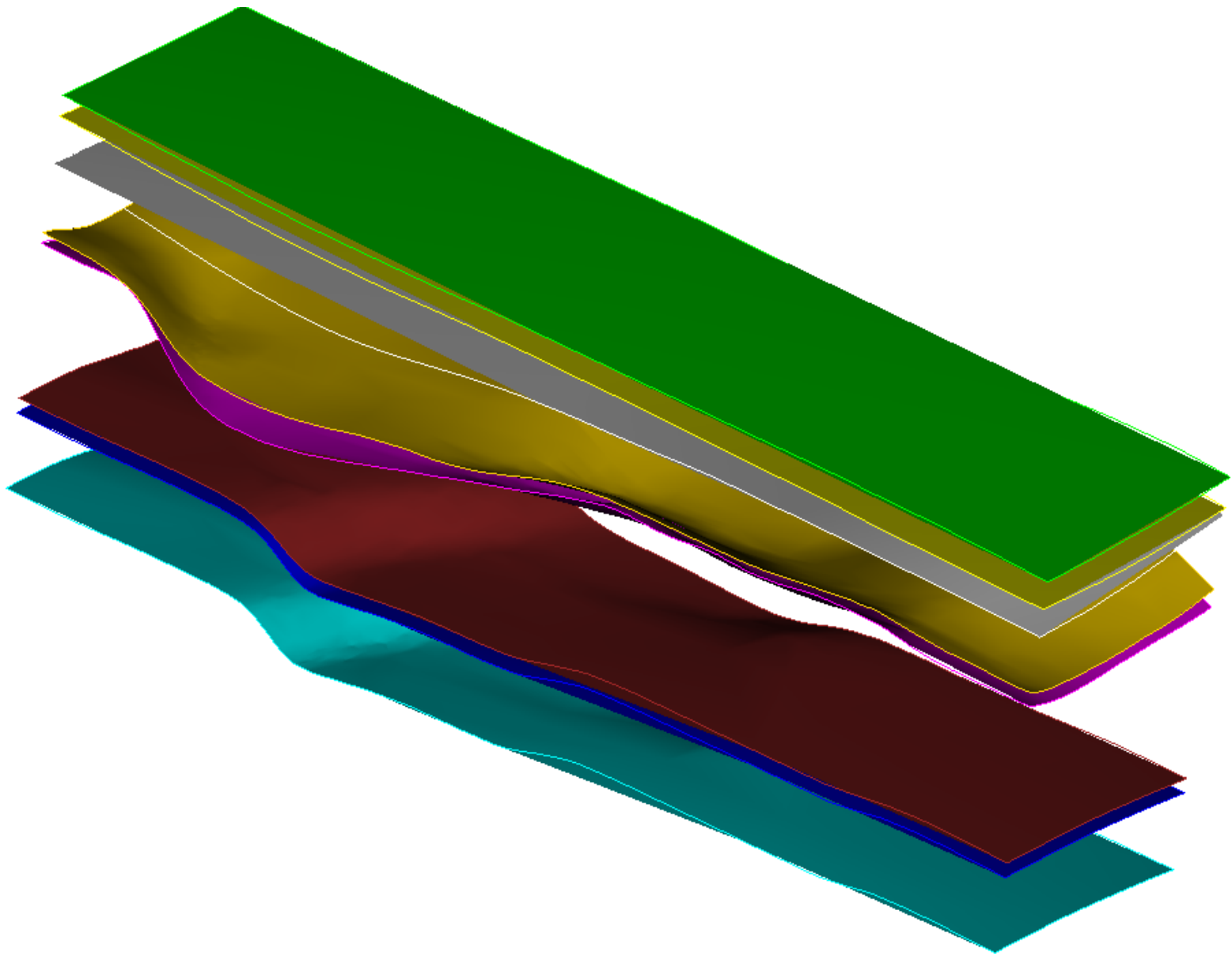


# MeshAssist

v1.0.0



***An open-source and cross-platform  
meshing assistant tool***

# MeshAssist v1.0.0

## User Manual

Hom Nath Gharti<sup>1</sup>, Princeton University, USA

Leah Langer, Princeton University, USA

Michael Roth, NORSAR, Norway

Jeroen Tromp, Princeton University, USA

Uno Vaaland, Princeton University, USA

Zhenzhen Yan, Institute of Remote Sensing and Digital Earth, CAS, China

November 28, 2017

<sup>1</sup>formerly at: NORSAR, Norway; and Institute of Engineering, Tribhuvan University, Nepal

# Licensing

MeshAssist v1.0.0 is free software: you can redistribute it and/or modify it under the terms of the GNU General Public License as published by the Free Software Foundation, either version 3 of the License, or (at your option) any later version.

MeshAssist v1.0.0 is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License for more details.

You should have received a copy of the GNU General Public License 3.0 along with MeshAssist v1.0.0. If not, see <<http://www.gnu.org/licenses/>>.

# Acknowledgments

Part of this document was prepared by using the documentation generator "Doxygen" (Main developer: Dimitri van Heesch) and the Matlab parser for Doxygen "doxymatlab" (Main developer: Fabrice).

# Contents

<b>Licensing</b>	<b>i</b>
<b>Acknowledgments</b>	<b>ii</b>
<b>1 Introduction</b>	<b>1</b>
1.1 Background	1
1.2 Cite as	1
1.3 Status summary	1
1.4 Changes made since the last version	1
<b>2 Getting started</b>	<b>2</b>
2.1 Package structure	2
2.2 Prerequisites	2
2.3 Configuration	3
2.4 Compile	3
2.5 Run	3
2.6 Bug Report	3
<b>3 File Index</b>	<b>4</b>
3.1 File List	4
<b>4 File Documentation</b>	<b>5</b>
4.1 dem2vti.m File Reference	5
4.1.1 Detailed Description	5
4.2 dxf2jou.m File Reference	6
4.2.1 Detailed Description	6
4.3 exodus2semgeotech.c File Reference	7
4.3.1 Detailed Description	7
4.4 exodus2specfem2d.c File Reference	9
4.4.1 Detailed Description	9
4.5 exodus2specfem3d.c File Reference	12
4.5.1 Detailed Description	12
4.6 exodusold2semgeotech.c File Reference	15
4.6.1 Detailed Description	15
4.7 exodusold2specfem3d.c File Reference	17
4.7.1 Detailed Description	17
4.8 gid2semgeotech.c File Reference	19
4.8.1 Detailed Description	19
4.9 gocad2vtu.c File Reference	21

4.9.1	Detailed Description . . . . .	21
4.10	vti2cell.c File Reference . . . . .	22
4.10.1	Detailed Description . . . . .	22
4.11	vtk1d2jou.f90 File Reference . . . . .	24
4.11.1	Detailed Description . . . . .	24
4.12	vtk2d2jou.c File Reference . . . . .	25
4.12.1	Detailed Description . . . . .	25
4.13	write_vti.m File Reference . . . . .	26
4.13.1	Detailed Description . . . . .	26
4.14	xyz2jou.f90 File Reference . . . . .	27
4.14.1	Detailed Description . . . . .	27

# Chapter 1

## Introduction

### 1.1 Background

MeshAssist is a collection of tools which assists meshing of complex and realistic 2D/3D models for FEM/SPECFEM simulations. As its name suggests, it is NOT a meshing software. It is only a meshing assistant!

### 1.2 Cite as

Gharti, H. N.; Langer, L.; Roth, M.; Tromp, J.; Vaaland, U.; Yan, Z. (2017). MeshAssist: an open-source and cross-platform meshing assistant tool. Zenodo. <http://doi.org/10.5281/zenodo.883448>

### 1.3 Status summary

Digital Elevation Model (DEM) : Yes

DXF AUTOCAD Model : Yes

GOCAD Model : Yes

EXODUS mesh : Yes

GiD mesh : Yes

VTK file : Yes

VTU file : Yes

XYZ file : Yes

### 1.4 Changes made since the last version

- This is the first version.

# Chapter 2

## Getting started

### 2.1 Package structure

The MeshAssist package can be obtained using Git. Use the following command in the terminal:

```
git clone --recursive https://github.com/homnath/MeshAssist.git
```

For the development version use the following command:

```
git clone --recursive --branch devel https://github.com/homnath/MeshAssist.git
```

The package has the following structure:

MeshAssist/

LICENSE	: License.
Makefile	: brief description of the package.
bin/	: all object files and executables are stored in this folder.
doc/	: documentation file/s for the MeshAssist package.
input/	: contains input files.
output/	: default output folder. All output files are stored in this folder unless the different output path is defined from command.
src/	: contains all source files.

### 2.2 Prerequisites

The package requires Make utility, latest C and Fortran compilers. For matlab files, Matlab is necessary.



## 2.3 Configuration

Open src/Makefile and modify the C and Fortran compilers if necessary.

## 2.4 Compile

Type the following command in the terminal

`make all`

Matlab files can be opened in and run from Matlab.

## 2.5 Run

*command input\_file [Options]*

Example:

`./bin/xyz2jou ./input/xyz2jou_example.utm`

See Chapter "File Documentation" for all available commands.

## 2.6 Bug Report

hgharti\_AT\_princeton\_DOT\_edu

# Chapter 3

## File Index

### 3.1 File List

Here is a list of all documented files with brief descriptions:

<a href="#">dem2vti.m</a>	Converts DEM image file to ASCII XYZ file . . . . .	5
<a href="#">dxf2jou.m</a>	Converts DXF AUTOCAD file to CUBIT/Trelis journal file . . . . .	6
<a href="#">exodus2semgeotech.c</a>	Converts ASCII exodus file to SPECFEM3D_GEOTECH files . . . . .	7
<a href="#">exodus2specfem2d.c</a>	Convert ASCII exodus file to SPECFEM2D format . . . . .	9
<a href="#">exodus2specfem3d.c</a>	Converts ASCII exodus file to SPECFEM3D files . . . . .	12
<a href="#">exodusold2semgeotech.c</a>	Converts old ASCII exodus file to SPECFEM3D_GEOTECH files . . . . .	15
<a href="#">exodusold2specfem3d.c</a>	Converts old ASCII exodus file to SPECFEM3D files . . . . .	17
<a href="#">gid2semgeotech.c</a>	Converts ASCII Gid mesh file to SPECFEM3D_GEOTECH files . . . . .	19
<a href="#">gocad2vtu.c</a>	Converts GOCAD ASCII file to VTU file . . . . .	21
<a href="#">vti2cell.c</a>	This file converts VTI file to VTU file . . . . .	22
<a href="#">vtk1d2jou.f90</a>	Converts VTK 1D file to CUBIT/Trelis journal file . . . . .	24
<a href="#">vtk2d2jou.c</a>	Converts VTK file consisting of 2D mesh to CUBIT/Trelis journal file . . . . .	25
<a href="#">write_vti.m</a>	Writes 3D gridded data to VTK VTI file . . . . .	26
<a href="#">xyz2jou.f90</a>	Converts UTM/XYZ file to CUBIT/Trelis journal file . . . . .	27

# Chapter 4

## File Documentation

### 4.1 dem2vti.m File Reference

Converts DEM image file to ASCII XYZ file.

#### 4.1.1 Detailed Description

Converts DEM image file to ASCII XYZ file.

This program converts DEM map (TIFF image format) to ASCII XYZ file and optionally ParaView/VTK VTI file format according to the parameters defined in the input file.

Note: Choosing a relatively small sampling interval may freeze the program due to the 'surfl' function.

Usage:

```
dem2vti(input_file, [output_path])
```

Example:

```
dem2vti('dem2vti_example.in')
```

OR

```
dem2vti('dem2vti_example.in', '../output')
```

Input:

*input\_file*: Name of DEM file.

Options:

An optional argument which must be a legitimate path can be provided as the output path. The default path is the current path.

Output:

All output files will be saved in the *output\_path* provided. If no *output\_path* is provided, the current path is used.

## 4.2 dxf2jou.m File Reference

Converts DXF AUTOCAD file to CUBIT/Trelis journal file.

### 4.2.1 Detailed Description

Converts DXF AUTOCAD file to CUBIT/Trelis journal file.

This function converts AUTOCAD 2000 (other ?) DXF ASCII file to a CUBIT/Trelis journal file, and optionally to a ParaView/VTK VTU ASCII file — an unstructured mesh file (.vtu). The function extracts only the faces represented by 'AcDbFace' tokens in the DXF file.

Usage:

```
dxf2jou(input_file, [save_vtu])
```

Example:

```
dxf2jou('../input/dxf2jou_example.dxf')
```

OR

```
dxf2jou('../input/dxf2jou_example.dxf',1)
```

Input:

`input_file`: DXF input file name

Options:

An optional argument can be provided to save VTU ASCII file (0: No [DEFAULT] or 1: yes)

.

Output:

VTK .vtu file which can be visualized with ParaView/VTK

## 4.3 exodus2semgeotech.c File Reference

Converts ASCII exodus file to SPECFEM3D\_GEOTECH files.

### Functions

- void **removeExtension** (char \*, char \*)
- int **get\_int** (int \*, char \*, char \*)
- int **look\_int** (int \*, char \*, char \*)
- int **getfirstquote** (char \*, char \*)
- int **main** (int argc, char \*\*argv)

### 4.3.1 Detailed Description

Converts ASCII exodus file to SPECFEM3D\_GEOTECH files.

This program converts the Binary (provided that "ncdump" command exists) or ASCII exodus file exported from CUBIT/Trelis to several mesh files required by the SPECFEM3D\_GEOTECH package. The "ncdump" command is a part of NetCDF library which is generally installed already in LINUX. If it is not installed, it can be downloaded for free from <https://www.unidata.ucar.edu/software/netcdf/>

Dependencies:

stringmanip.c: string manipulation routines

Compile:

```
gcc exodus2semgeotech.c -o exodus2semgeotech
```

Usage:

```
exodus2semgeotech input_file [Options]
```

Example:

```
exodus2semgeotech tunnel.e -bin=1
```

or

```
exodus2semgeotech tunnel.txt
```

Options:

- -fac: Use this option to multiply coordinates by some factor. This is important for unit conversion, e.g., to convert m to km use -fac=0.001 [DEFAULT 1]
- -bin: Use this option if you want to convert exodus binary file directly, provided that the command "ncdump" is in the path. The command "ncdump" is a part of netCDF library that can be downloaded for free from <http://www.unidata.ucar.edu/downloads/netcdf/index.jsp>. Use -bin=1 for binary or -bin=0 for ascii file. [DEFAULT 0]

Issues:

- - This does not work with older versions of CUBIT. For the older versions use [exodusold2semgeotech.c](#).

Basic steps starting from CUBIT/TRELIS:

step 1: prepare mesh in TRELIS/CUBIT

- Define material regions using "Blocks"

For example:

block 1 add volume 1

block 2 add volume 2 3

will assign material region 1 to volume 1 and material region 2 to volumes 2 and 3. These material regions will be used to define material properties in "\*material\_list". This program will NOT generate "\*material\_list". The file "material\_list" must be created to run SPEC3D\_GEOTECH!

- Define surface boundary conditions using "Nodesets" or "Sidesets" – nodal boundary conditions must be defined using node set -> each node set name must contain the corresponding BC names as defined in char \*ns\_bcname[] below  
e.g., node set name can be front\_nsbucx or front\_nsbucx\_nsbucy etc. – surface boundary conditions must be defined using side set -> each side set name must contain the corresponding BC names as defined in char \*ss\_bcname[] below  
e.g., side set name can be front\_ssbucx or front\_ssbucx\_ssbucy etc.

For example:

sideset 1 add surface 1

sideset 1 name 'bottom\_ssbucx\_ssbucy\_ssbucz'

will define a surface in which all displacement components are prescribed.

Note: All the above commands can also be executed using TRELIS/CUBIT GUI. "sideset 1 name 'bottom\_ssbucx\_ssbucy\_ssbucz'" is equivalent to clicking 'sideset 1' and renaming.

step2: export mesh file as exodus file say "tunnel.e" (use 3D option)

step3: convert "tunnel.e" to SPEC3D files `exodus2semgeotech tunnel.e -bin=1`

There will be several output files:

- \*\_coord\_? : coordinates file => total number of nodes followed by nodal coordinate ? (? -> x, y, z)
- \*\_connectivity : element file => total number of elements followed by connectivity list
- \*\_material\_id : material file => total number of elements followed by material IDs
- \*\_??bcu? : node IDs which have u? defined as the boundary conditions (?? -> ns or ss, ? -> x, y, z). Total number of entities (nodes or faces) followed by element ID and surface nodes.

## 4.4 exodus2specfem2d.c File Reference

Convert ASCII exodus file to SPECFEM2D format.

### Functions

- void **removeExtension** (char \*, char \*)
- int **get\_int** (int \*, char \*, char \*)
- int **look\_int** (int \*, char \*, char \*)
- int **getfirstquote** (char \*, char \*)
- int **shape** (double, double, double \*\*)
- int **check\_normal** (double [3][4], double [3])
- int **isclockwise** (int, double [], double [])
- int **main** (int argc, char \*\*argv)
- int **isclockwise** (int n, double x[n], double z[n])

### 4.4.1 Detailed Description

Convert ASCII exodus file to SPECFEM2D format.

This program converts the ASCII exodus file exported from CUBIT to several input files required by the SPECFEM2D program. Currently, this program only handles the 2D quadrilateral elements with four nodes. The binary exodus file (e.g., .e file) needs to be converted into an ASCII file, generally using a free console application "ncdump" which is a part of the netCDF library, and can be downloaded from <http://www.unidata.ucar.edu/downloads/netcdf/index.jsp>. Please see the detailed steps below.

Dependencies:

stringmanip.c: string manipulation routines

Compile:

```
gcc exodus2specfem2d.c -o exodus2specfem2d -lm
```

Usage:

```
exodus2specfem2d input_file [Options]
```

Example:

```
exodus2specfem2d mesh.e -bin=1
```

or

```
exodus2specfem2d mesh.txt
```

## Options:

- -fac: Use this option to multiply coordinates by some factor. This is important for unit conversion, e.g., to convert m to km use -fac=0.001 [DEFAULT 1]
- -bin: Use this option if you want to convert exodus binary directly, provided that the command "ncdump" is in the path. The command "ncdump" is a part of netCDF library that can be downloaded for free from <http://www.unidata.ucar.edu/downloads/netcdf/index.jsp>. Use -bin=1 for binary or -bin=0 for ascii file. [DEFAULT 0]
- -order: Use this option to check the connectivity order and make sure that the connectivity is in counterclockwise order. Use -order=1 for checking or -order=0 for no checking [DEFAULT 0].
- -head: Use this option to attach head of input file to output file names. Use -head=1 to attach header or -head=0 not to attach [DEFAULT 0]
- -tomo: Use this option for tomography model. Since tomography model uses negative identifiers, this option will write negative block IDs. Use -tomo=1 to make negative block IDs or -tomo=0 not to make [DEFAULT 0]

## Basic steps starting from TRELIS:

### Step 1: prepare mesh in TRELIS/CUBIT

- Define material regions using "Blocks"

For example:

```
block 1 add surface 1
```

```
block 2 add surface 2 3
```

will assign material region 1 to surface 1 and material region 2 to surfaces 2 and 3. These material regions will be used to define material properties in "Par\_file". This program will NOT generate "Par\_file". The file "Par\_file" must be created to run SPECDEM2D!

- Define element type to be QUAD4

For example:

```
block all element type quad4
```

NOTE: If the element types are SHELL or SHELL4, "Default" or 3D option should be selected during export. If the element type is QUAD or QUAD4, 3D option should be selected. With default or 2D data, it saves only X and Y coordinates which is not always correct. Make sure that the node ordering is strictly anticlockwise (no longer necessary!) for all the elements in CUBIT.

- Define surface boundary conditions using "Sidesets"

For example:

```
sideset 1 add curve 1
```

```
sideset 1 name 'free_surface_file'
```



will define a free or absorbing surface boundary condition on surface. Similarly,  
sideset 2 add curve 3  
sideset 2 name 'absorbing\_surface\_file'

will define absorbing boundary condition on the curve 3. Note: All the above commands can also be executed using TRELIS/CUBIT GUI. "sideset 1 name 'free\_surface\_file'" is equivalent to clicking sideset 1 and renaming.

Step 2: export mesh file as exodus file say "mesh.e" (always use 3D option!)

Step 3: convert "mesh.e" to SPECFEM2D files `exodus2specfem2d mesh.e -bin=1`

There will be several output files:

- coordinates : coordinates file => total number of nodes followed by nodal coordinate ? (? -> x, y, z)
- connectivity : element file => total number of elements followed by connectivity list
- materials : material file => total number of elements followed by material IDs
- surface\* : surface boundary condition files => total number of elements followed by element ID and surface nodes

## 4.5 exodus2specfem3d.c File Reference

Converts ASCII exodus file to SPECFEM3D files.

### Functions

- void **removeExtension** (char \*, char \*)
- int **get\_int** (int \*, char \*, char \*)
- int **look\_int** (int \*, char \*, char \*)
- int **getfirstquote** (char \*, char \*)
- int **check\_normal** (double [3][4], double [3])
- int **main** (int argc, char \*\*argv)

### 4.5.1 Detailed Description

Converts ASCII exodus file to SPECFEM3D files.

This program converts the Binary (provided that "ncdump" command exists, type "ncdump" to check whether "ncdump" command exists.) or ASCII exodus file exported from TRELIS/↔ CUBIT to several mesh files required by the SPECFEM3D Cartesian package. The "ncdump" command is a part of NetCDF library which is generally installed already in LINUX. If this library is not found, it can be downloaded for free from

<https://www.unidata.ucar.edu/software/netcdf/>

Dependencies:

stringmanip.c: string manipulation routines

Compile:

gcc [exodus2specfem3d.c](#) -o exodus2specfem3d

Usage:

exodus2specfem3d *input\_file* [*Options*]

Example:

exodus2specfem3d tunnel.e -bin=1

or

exodus2specfem3d tunnel.txt

Options:

- -fac: Use this option to multiply coordinates by some factor. This is important for unit conversion, e.g., to convert m to km use -fac=0.001 [DEFAULT 1]
- -bin: Use this option if you want to convert exodus binary directly, provided that the command "ncdump" is in the path. The command "ncdump" is a part of netCDF library that can be downloaded for free from <http://www.unidata.ucar.edu/downloads/netcdf/index.jsp>. Use -bin=1 for binary or -bin=0 for ascii file. [DEFAULT 0]

- -norm: Use this option to check the normal of the faces in order to make sure that the surface nodes are in the right order. Use -norm=1 for checking or -norm=0 for no checking [DEFAULT 0]. Normally this is not necessary.
- -head: Use this option to attach head of input file to output file names. Use -head=1 to attach header or -head=0 not to attach [DEFAULT 0]
- -tomo: Use this option for tomography model. Since tomography model uses negative identifiers, this option will write negative block IDs. Use -tomo=1 to make negative block IDs or -tomo=0 not to make [DEFAULT 0]

Issues:

- - This does not work with older version of CUBIT. For the older version use [exodusold2specfem3d.c](#).

Basic steps starting from the TRELIS:

step 1: prepare mesh in TRELIS/CUBIT

- Define material regions using "Blocks"

For example:

block 1 add volume 1

block 2 add volume 2 3

will assign material region 1 to volume 1 and material region 2 to volumes 2 and 3. These material regions will be used to define material properties in "nummaterial\_↵ velocity\_file". This program will NOT generate "nummaterial\_velocity\_file". The file "nummaerial\_veolicty\_file" must be created to run SPECSEM3D!

- Define surface boundary conditions using "Sidesets"

For example:

sideset 1 add surface 1

sideset 1 name 'free\_or\_absorbing\_surface\_file\_zmax'

will define a free or absorbing surface boundary condition on surface 1 which lies at the top of the volume (zmax). similarly,

sideset 2 add surface 3

sideset 2 name 'absorbing\_surface\_file\_bottom'

will define absorbing boundary condition on the surface 3 which lies at the bottom of the volume (zmin). Note: All the above commands can also be executed using TRELIS/CUBIT GUI. "sideset 1 name 'free\_or\_absorbing\_surface\_file\_zmax'" is equivalent to clicking sideset 1 and renaming.

step2: export mesh file as exodus file say "tunnel.e" (use 3D option)

step3: convert "tunnel.e" to SPECSEM3D files `exodus2specsem3d tunnel.e -bin=1`

There will be several output files:

- `nodes_coords_file` : coordinates file => total number of nodes followed by nodal coordinate ? (? -> x, y, z)
- `mesh_file` : element file => total number of elements followed by connectivity list
- `materials_file` : material file => total number of elements followed by material IDs
- `surface_file*` : surface boundary condition files => total number of elements followed by element ID and surface nodes

## 4.6 exodusold2semgeotech.c File Reference

Converts old ASCII exodus file to SPECFEM3D\_GEOTECH files.

### Functions

- void **removeExtension** (char \*, char \*)
- int **get\_int** (int \*, char \*, char \*)
- int **look\_int** (int \*, char \*, char \*)
- int **getfirstquote** (char \*, char \*)
- int **main** (int argc, char \*\*argv)

### 4.6.1 Detailed Description

Converts old ASCII exodus file to SPECFEM3D\_GEOTECH files.

This program converts the Binary (provided that "ncdump" command exists) or ASCII exodus file exported from the old CUBIT to several mesh files required by the SPECFEM3D\_GEOTECH package. The "ncdump" command is a part of NetCDF library which is generally installed already in LINUX, which can be downloaded for free from

<https://www.unidata.ucar.edu/software/netcdf/>

Dependencies:

stringmanip.c: string manipulation routines

Compile:

gcc [exodusold2semgeotech.c](#) -o exodusold2semgeotech

Usage:

exodusold2semgeotech *input\_file* [*Options*]

Example:

exodusold2semgeotech tunnel.e -bin=1

or

exodusold2semgeotech tunnel.txt

Options:

- -fac: Use this option to multiply coordinates by some factor. This is important for unit conversion, e.g., to convert m to km use -fac=0.001 [DEFAULT 1]
- -bin: Use this option if you want to convert exodus binary directly, provided that the command "ncdump" is in the path. The command "ncdump" is a part of netCDF library that can be downloaded for free from <http://www.unidata.ucar.edu/downloads/netcdf/index.jsp>. Use -bin=1 for binary or -bin=0 for ascii file. [DEFAULT 0]

Issues:

- - This does not work with older version of CUBIT. For the older version use [exodusold2semgeotech.c](#).

Basic steps starting from the CUBIT:

step 1: prepare mesh in CUBIT

- Define material regions using "Blocks"

For example:

block 1 add volume 1

block 2 add volume 2 3

will assign material region 1 to volume 1 and material region 2 to volumes 2 and 3. These material regions will be used to define material properties in "nummaterial\_↔ velocity\_file". This program will NOT generate "nummaterial\_velocity\_file". The file "nummaerial\_veolicty\_file" must be created to run SPECSEM3D!

- Define surface boundary conditions using "Nodesets" or "Sidesets" – nodal boundary conditions must be defined using node set -> each node set name must contain the corresponding BC names as defined in char \*ns\_bcname[] below  
e.g., node set name can be front\_nsbucx or front\_nsbucx\_nsbucy etc. – surface boundary conditions must be defined using side set -> each side set name must contain the corresponding BC names as defined in char \*ss\_bcname[] below  
e.g., side set name can be front\_ssbucx or front\_ssbucx\_ssbucy etc.

For example:

sideset 1 add surface 1

sideset 1 name 'bottom\_ssbucx\_ssbucy\_ssbucz'

will define a surface in which all displacement components are prescribed.

Note: All the above commands can also be executed using TRELIS/CUBIT GUI. "sideset 1 name 'bottom\_ssbucx\_ssbucy\_ssbucz'" is equivalent to clicking 'sideset 1' and renaming.

step2: export mesh file as exodus file say "tunnel.e" (use 3D option)

step3: convert "tunnel.e" to SPECSEM3D files `exodusold2semgeotech tunnel.e -bin=1`

There will be several output files:

- \*\_coord\_? : coordinates file => total number of nodes followed by nodal coordinate ? (? -> x, y, z)
- \*\_connectivity : element file => total number of elements followed by connectivity list
- \*\_material\_id : material file => total number of elements followed by material IDs
- \*\_??bcu? : node IDs which have u? defined as the boundary conditions (?? -> ns or ss, ? -> x, y, z). Total number of entities (nodes or faces) followed by element ID and surface nodes.

## 4.7 exodusold2specfem3d.c File Reference

Converts old ASCII exodus file to SPECFEM3D files.

### Functions

- void **removeExtension** (char \*, char \*)
- int **get\_int** (int \*, char \*, char \*)
- int **look\_int** (int \*, char \*, char \*)
- int **getfirstquote** (char \*, char \*)
- int **check\_normal** (double [3][4], double [3])
- int **main** (int argc, char \*\*argv)

### 4.7.1 Detailed Description

Converts old ASCII exodus file to SPECFEM3D files.

This program converts the Binary (provided that "ncdump" command exists) or ASCII exodus file exported from the old CUBIT to several mesh files required by the SPECFEM3D package.

Dependencies:

stringmanip.c: string manipulation routines

Compile:

```
gcc exodusold2specfem3d.c -o exodusold2specfem3d
```

Usage:

```
exodusold2specfem3d input_file [Options]
```

Example: exodusold2specfem3d tunnel.txt

or

```
exodusold2specfem3d tunnel.e -fac=0.001 -bin=1
```

Options:

- -fac: use this option to multiply coordinates. this is important for unit conversion, e.g., to convert m to km use -fac=0.001
- -bin: use this option if you want to convert exodus binary directly, provided that the command "ncdump" is in the path. The command "ncdump" is a part of netCDF library that can be downloaded for free from <http://www.unidata.ucar.edu/downloads/netcdf/index.jsp>. use -bin=1 for binary or -bin=0 for ascii file.
- -norm: use this option to check the normal of the faces. use -norm=1 for checking or -norm=0 (default) for no checking

Issues:

- - This does not work with new version of Trelis/CUBIT. For the new version use [exodus2specfem3d.c](#).

# Basic steps starting from the CUBIT:

step 1: prepare mesh in CUBIT

- define material regions using "Blocks"

For example:

block 1 add volume 1

block 2 add volume 2 3

will assign material region 1 to volume 1 and material region 2 to volumes 2 and 3. These material regions will be used to define material properties in "nummaterial\_↔velocity\_file". this program will NOT generate "nummaterial\_velocity\_file". the file "nummaerial\_veolicty\_file" must be created to run SPECSEM3D!

define surface boundary conditions using "Sidesets"

For example:

sideset 1 add surface 1

sideset 1 name 'free\_or\_absorbing\_surface\_file\_zmax'

will define a free or absorbing surface boundary condition on surface 1 which lies at the top of the volume (zmax). similarly,

sideset 2 add surface 3

sideset 2 name 'absorbing\_surface\_file\_bottom'

will define absorbing boundary condition on the surface 3 which lies at the bottom of the volume (zmin). Note: All the above commands can also be executed using TRELIS/CUB↔IT GUI. "sideset 1 name 'free\_or\_absorbing\_surface\_file\_zmax'" is equivalent to clicking sideset 1 and renaming.

step2: export mesh file as exodus file say "tunnel.e" (use 3D option)

step3: convert "tunnel.e" to SPECSEM3D files `exodusold2specfem3d tunnel.e -bin=1`

There will be several output files:

- nodes\_coords\_file : coordinates file => total number of nodes followed by nodal coordinate ? (? -> x, y, z)
- mesh\_file : element file => total number of elements followed by connectivity list
- materials\_file : material file => total number of elements followed by material IDs
- surface\_file\* : surface boundary condition files => total number of elements followed by element ID and surface nodes



## 4.8 gid2semgeotech.c File Reference

Converts ASCII Gid mesh file to SPECFEM3D\_GEOTECH files.

### Functions

- void **removeExtension** (char \*, char \*)
- int **get\_int** (int \*, char \*, char \*)
- int **look\_int** (int \*, char \*, char \*)
- int **getfirstquote** (char \*, char \*)
- int **main** (int argc, char \*\*argv)

### 4.8.1 Detailed Description

Converts ASCII Gid mesh file to SPECFEM3D\_GEOTECH files.

This program converts the ASCII GiD mesh file to several mesh files required by the SPECFEM3D\_GEOTECH package. GiD ([www.gidhome.com](http://www.gidhome.com)) is a commercial pre and post processor for numerical simulations.

Dependencies:

stringmanip.c: string manipulation routines

Compile:

```
gcc gid2semgeotech.c -o gid2semgeotech
```

Usage:

```
gid2semgeotech input_file [Options]
```

Example:

```
gid2semgeotech gid2semgeotech__example.dat
```

or

```
gid2semgeotech gid2semgeotech__example.dat -fac=0.001
```

Options:

- -fac: Use this option to multiply coordinates with a certain factor. This is useful for unit conversion, e.g., to convert m to km use: -fac=0.001

Basic steps starting from GID:

step1: Export mesh file in ASCII format "mesh.dat"

step2: Produce mesh and BC files   gid2semgeotech mesh.dat

OR

gid2semgeotech mesh.dat 1000.0

There will be several output files:

- coord\_? : Total number of nodes followed by nodal coordinate ? (? -> x,y,z)
- \_connectivity : Total number of elements followed by connectivity list
- \_material\_id : Total number of elements followed by material IDs
- ??bcu? : node IDs which have  $u? = 0$  as the boundary conditions (?? -> ns or ss, ? -> x,y,z)

## 4.9 gocad2vtu.c File Reference

Converts GOCAD ASCII file to VTU file.

### Functions

- `int main (int argc, char **argv)`

### 4.9.1 Detailed Description

Converts GOCAD ASCII file to VTU file.

This program converts the GOCAD ASCII file (3-noded triangular meshes) to VTK XML .vtu binary file (unstructured mesh file) which can be visualized/processed in ParaView or VTK.

Dependencies:

stringmanip.c

Compile

- in parent folder, type: `make` OR
- in `src/` folder, type `gcc gocad2vtu.c -o gocad2vtu`

Usage:

`./bin/gocad2vtu input_file [Options]`

Example: `./bin/gocad2vtu ./input/gocad2vtu_example.ts`

Options:

- `-fac`: Use this option to multiply the coordinates by a certain factor, this is helpful for unit conversion, e.g. for m to km use 0.001, for km to m use 1000, example: `gocad2vtu T2_horizon.ts -fac=0.001`

Notes:

- Output .vtu file is binary, therefore endianness of the processor architecture is important.
- This program automatically identify the endianness and write the output accordingly. Hence if you run and process/visualize .vtu file in the architecture with different endianness there may be an error.

## 4.10 vti2cell.c File Reference

This file converts VTI file to VTU file.

### Functions

- int **comp\_float** (const void \*a, const void \*b)
- int **main** (int argc, char \*\*argv)

### 4.10.1 Detailed Description

This file converts VTI file to VTU file.

This program converts the 2D/3D Binary VTK XML .vti file to unstructured mesh files (.vtu). This program also generates the mesh files required by SPECSEM2D and SPECSEM3D. Note that the file formats in SPECSEM2D and SPECSEM3D are different. This should be made same format as soon as possible. For this, source codes within the decompose folder of SPECSEM3D and cubit2specsem3d.py need to be changed.

Dependencies:

stringmanip.c

Compile:

```
gcc vti2cell.c -o vti2cell -lm
```

Usage:

```
vti2cell input_file [Options]
```

Example:

```
vti2cell py_plane_model.vti
```

Options:

- -fac=factor (real) Use this option to multiply the coordinates by a certain factor, this is helpful for unit conversion, e.g. for m to km use 0.001, for km to m use 1000 Example: vti2cell2d py\_plane\_model.vti -fac=1000
- -xmat=exclusion material id/s (integer/s) Use this option to exclude certain region of the model, e.g. exclusion of air. Appropriate id/s should be supplied, id s are number ordered according to the value of corresponding material properties and numbered starting from 1. This way, lowest value will have id 1 and so on. Example: vti2cell py\_plane\_model.vti -xmat=1,2 This command will exclude the regions with material id 1 and 2.

Example: vti2cell py\_plane\_model.vti -fac=1000 -xmat=1 This command multiply the coordinates by 1000 and exclude the region with material id 1

- -step=step size (integer) Use this option to coarsen the mesh. This value represent the number of grids to be used as 1 element, e.g., if you want to make 2 grids as 1 element, use -step=2
- -zup=z axis direction indicator (integer) Use this option to indicate whether the Z axis direction is up

Toto:

- make uniformity for 2D,3D, e.g., writing and reading coordinates

## 4.11 vtk1d2jou.f90 File Reference

Converts VTK 1D file to CUBIT/Trelis journal file.

Functions/Subroutines

- program **vtk1d2jou**

### 4.11.1 Detailed Description

Converts VTK 1D file to CUBIT/Trelis journal file.

This program reads ASCII vtk files with unstructured grid of lines and points only, and removes the redundant lines. The redundant nodes can be removed within the paraview itself using the 'Clean to Grid' filter.

Compile:

```
gfortran vtk1d2jou.f90 -o vtk1d2jou
```

Usage:

```
vtk1d2jou input_file
```

```
vtk1d2jou vtk1d2jou_example.vtk
```

## 4.12 vtk2d2jou.c File Reference

Converts VTK file consisting of 2D mesh to CUBIT/Trelis journal file.

### Functions

- void **removeExtension** (char \*, char \*)
- int **main** (int argc, char \*\*argv)

### 4.12.1 Detailed Description

Converts VTK file consisting of 2D mesh to CUBIT/Trelis journal file.

This program converts an ASCII VTK file consisting of triangular/quadrilateral mesh into a CUBIT/Trelis Journal file.

Dependences:

stringmanip.c: string manipulation routines

Compile:

```
gcc vtk2d2jou.c -o vtk2d2jou
```

Usage:

```
vtk2d2jou input_file
```

Example:

```
vtk2d2jou vtk2d2jou__example.vtk
```

## 4.13 write\_vti.m File Reference

Writes 3D gridded data to VTK VTI file.

### 4.13.1 Detailed Description

Writes 3D gridded data to VTK VTI file.

This function writes the VTI binary file for structured grid data, such as finite difference data and tomography data. The VTI file can be visualized in ParaView (<http://www.paraview.org/>).

Input:

fname : output file name

ox : origin vector [ox oy oz]

dh : sampling interval vector [dx dy dz]

nx : grid number vector [nx ny nz]

name : output variable name

Usage:

call this functions with appropriate variables.

Notes:

- For a BigEndian architecture, replace "LittleEndian" with "BigEndian" in the line below.



## 4.14 xyz2jou.f90 File Reference

Converts UTM/XYZ file to CUBIT/Trelis journal file.

Functions/Subroutines

- program **xyz2jou**

### 4.14.1 Detailed Description

Converts UTM/XYZ file to CUBIT/Trelis journal file.

This program converts a UTM or XYZ file to a CUBIT journal file. The UTM or XYZ file contains the three columns of X, Y, and Z coordinates, respectively.

Compile:

- in parent folder, type: `make OR`
- in `src/` folder, type `gfortran xyz2jou.f90 -o xyz2jou`

Usage:

`./bin/xyz2jou input_file [Options]`

Example:

`./bin/xyz2jou ./input/xyz2jou_example.utm`

Options:

- `-nx`: Use this option if you know the number of points in a line along X axis . This will speed up the processing. For example, `-nx=100`. If it is not defined, `nx` is automatically determined.
- `-nskip`: Use this option if you want to skip (downsample) certain number of successive points. This will skip along both X and Y axes. For example, `-nskip=2`. [DEFAULT 0].

## Bibliography