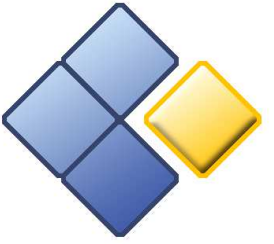
A Full 3-D Parallel EM Simulation Software and System

GEMS



User's Manual

Version 7.8

2COMU, Inc.
4031 University Dr.
Fairfax, VA 22030 USA
Web: http://www.2comu.com

Notice

The information contained in this document is subject to change without notice.

2COMU makes no warranty of any kind with regard to this material, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. 2COMU shall not be liable for simulation result errors and computer and network security contained herein or for incidental or consequential damages in connection with the furnishing, performance, or the use of this material.

2COMU, Inc.
4031 University Dr.
Fairfax, VA 22030 USA
Ph: (814) 441-9403
Fax: (703)277-7730

Table of Contents

Chapter 1 GEMS Installation

You need to receive GEMS CDs from 2COMU or download GEMS package from 2COMU server that is authorized by 2COMU. And then you can start to install GEMS package on your PC, workstation, PC cluster, or high performance cluster.

1.1 Install GEMS Package on PC

GEMS software package has 32-bit and 64-bit versions. GEMS 32-bit version can be only installed on a 32-bit platform, and likewise, GEMS 64-bit version can be only installed on a 64-bit platform. Both GEMS 32-bit and 64-bit versions support multiple core CPU computer and multiple CPU workstation, computer cluster. GEMS hardware acceleration and parallel processing modules are optional and will require the users to pay separately.

1.1.1 Installation Request

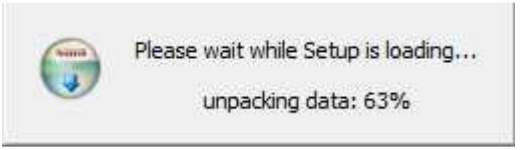
GEMS can only be installed by the administrator or the user with the administrative privileges. 2COMU requires the user registering to 2COMU before starting to use GEMS software package. GEMS registration requires the users to provide the basic computer information, and the license will be good through the licensed period.

1.1.2 Single Seat License

GEMS setup file provides a graphical step-by-step procedure to install GEMS package on your computer. After GEMS package is successfully installed on your computer, GEMS will create several shortcuts, Designer, Display, Solver, GEMS online, License manager and uninstall in the **Start** program for quick access to GEMS modules.

Follow the steps given below to install the serial version of GEMS:

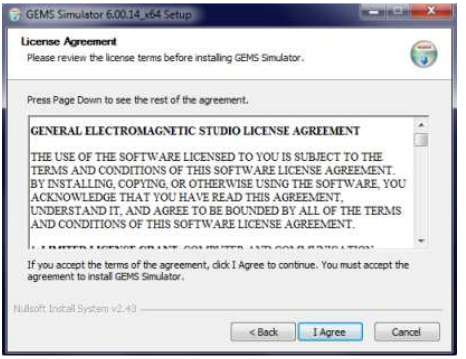
(1) Double-click on the **GEMS_Setup.exe** file and you will see the following message:



(2) Click on the **Yes** button to install the MPI library. You can click on the **No** button to skip this step if you installed GEMS package on your computer before.

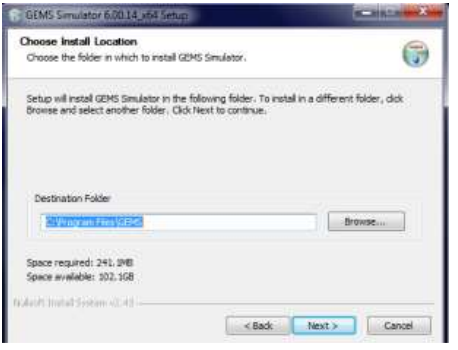
(3) Click on the **Next** button to start GEMS installation.

(4) Click on the **I Agree** button if you agree the license agreement.



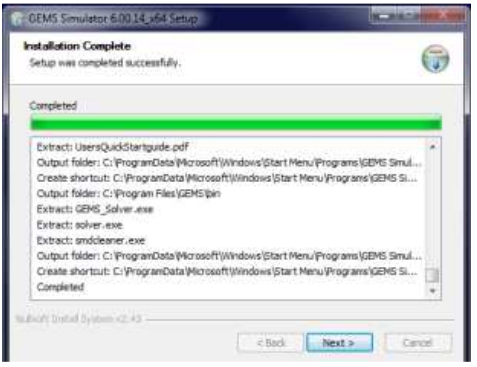
(5) Click on the **Next** button to install the default options.

(6) Select the installation folder and then click on the **Next** button.



(7) Choose the start menu folder and then click on the **Install** button.

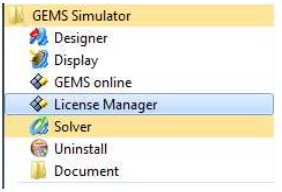
(8) Click on the **Next** button after the installation is completed.



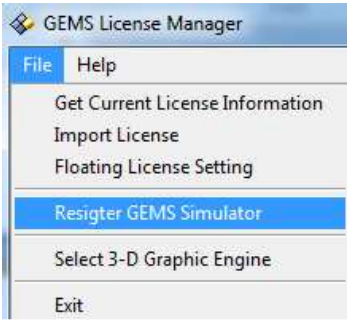
(9) Click on the **Finish** button to complete the installation.

(10) Register to 2COMU

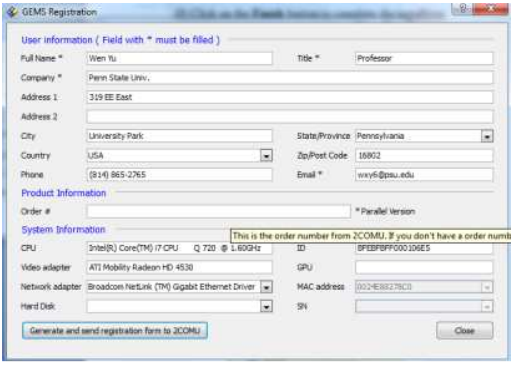
(i) Select the L**icense Manager** option in the **Start** program



(ii) Select the **Register GEMS Simulator** option in the **File** menu



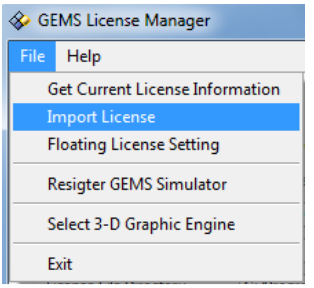
(iii) Fill the form out, save it in a text file and then email it to 2COMU.



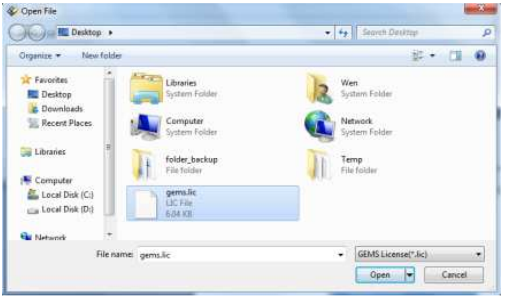
You will receive a license file from 2COMU in 48 hours.

(11) Import the license file you received from 2COMU.

(i) Select the **Import License** option in the **File** menu

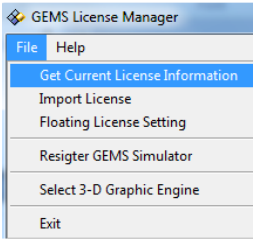


(ii) Search and select "gems.lic" file and then click on the **Open** button

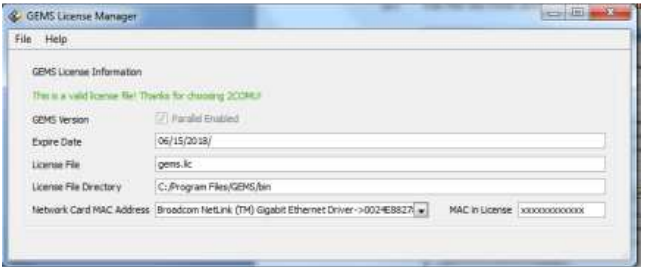


(12) Check license status

(i) Select the **Get Current License Information** option in the **File** menu.

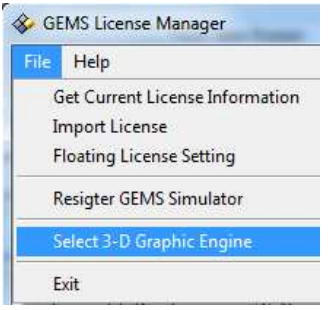


(ii) The license information is shown in the window below.

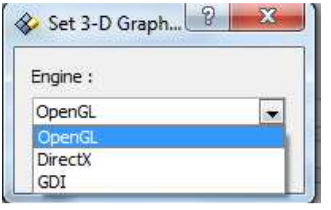


(13) Select the graphical support method

(i) Select the **Select 3-D Graphic Engine** option in the **File** menu



(ii) Select the OpenGL (default), DirectX, or GDI.



1.1.3 Floating License

GEMS floating license installation has the same procedure as the single seat license. However, you need a server computer for the floating license installation.

Server computer:

(i) Install the GEMS floating license on the server computer;

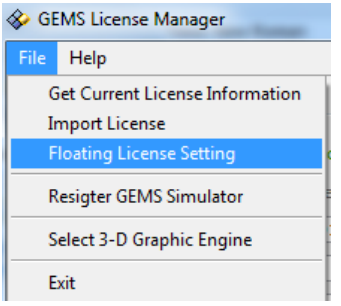
(ii) Register GEMS software on this server to 2COMU;

(iii) Import the GEMS license file you received from 2COMU.

Client computer:

(i) Install GEMS software (do not need to register GEMS).

(ii) Select the **Floating License Setting** option in the **File** menu.



(iii) Input the server IP.

1.2 Install GEMS Package on PC Cluster

GEMS PC cluster version requires that the users have two or more computers that have been properly connected each other through network system such as gigabit Ethernet, 10-gigabit Ethernet, Infiniband or Myrinet network. If the cluster includes only two PC computers, you can connect them through an internet cable. The computers in the PC cluster are better to have the same configurations to reach the better parallel performance. Follow the steps given below to install the GEMS on a two-computer cluster:

(1) Install GEMS package on each computer;

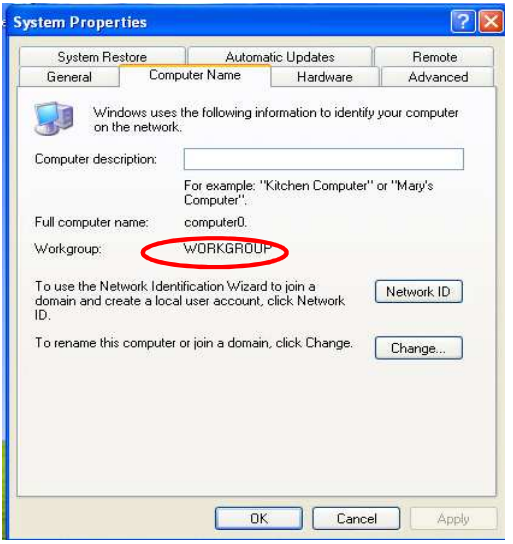
(2) Register each computer to 2COMU;

(3) Run a simple case on each computer and ensure GEMS works on both computers;

(4) Connect two computers through a network cable;

(5) Move the mouse icon to the computer in the **Start** program and press the right mouse button;

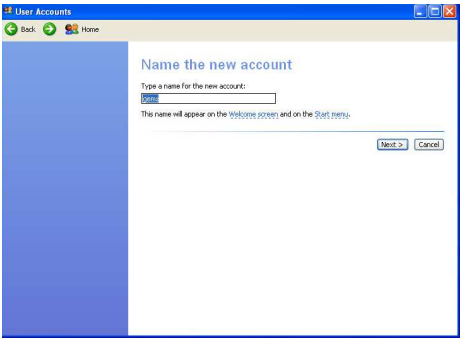
(6) Check the full computer name and Workgroup or Domain name;



(7) Ensure two computers in the same group or domain;

(8) Create a new user account in the administrator or an account with the same priority in each computer;

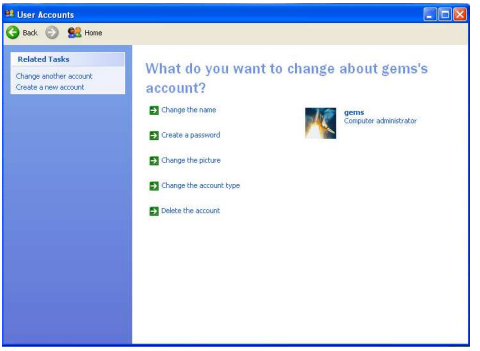
(9) The usernames and passwords should be same on two computers;



(10) Go to the "Control Panel" window and then to the "User Accounts" option;

(11) Select a user account;

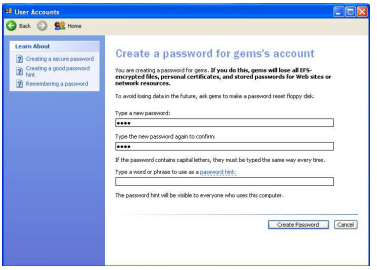
(12) Click on the "Create a password" link.



(13) Type a password;

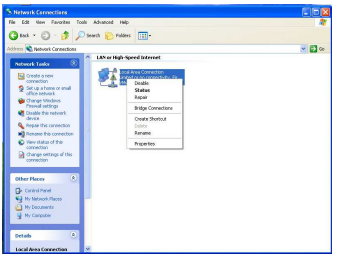
(14) Confirm the password;

(15) Repeat the same procedure for each computer;



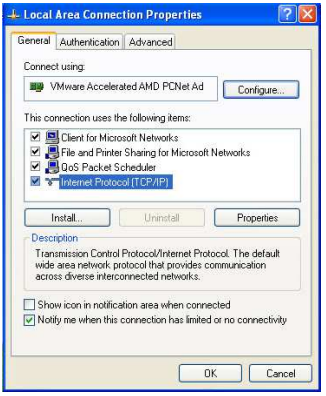
(16) Go to the "Control Panel" -> "Network and Internet Connections" -> "Network Connections;"

(17) Move the mouse icon to the "Local Area Connection" and click on the right mouse; button and then select "Properties;"

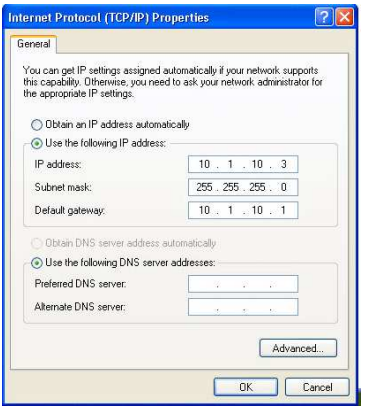


(18) Select "Internet Protocol (TCP/IP);

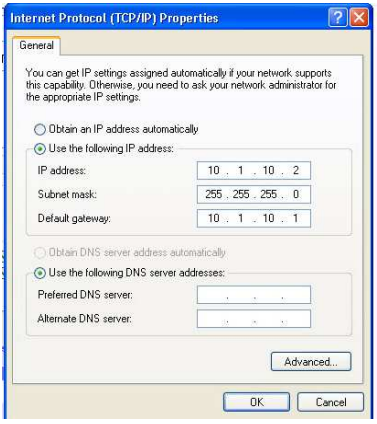
(19) Click on "Properties" button;



(20) For the computer one;



(21) For the computer two;

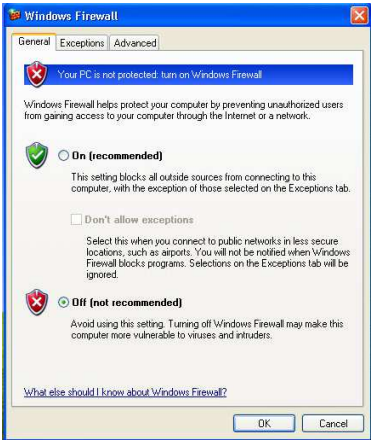


(22) Check the network connection;

(23) temporally disable Windows Firewall;

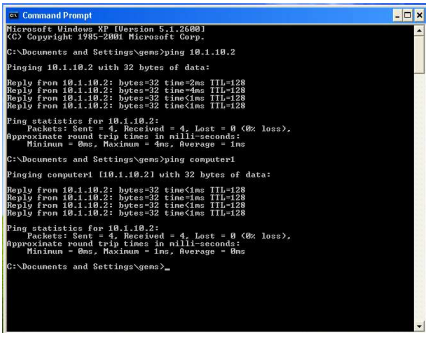
(24) Go to the "Control Panel"->"Security Center" -> "Windows Firewall";

(25) Click on "Off(not recommended)" Radio button;



(26) Go to "Start" -> "Run" option;

(27) Type "Ping" + computer name or IP address;



(28) Go to "Control Panel" and then click on the "Firewall settings" option;

(29) Add the generator.exe and GEMS_Solver.exe to Windows Firewall Exceptions list;

(30) Open Windows Firewall settings;

(31) Click on the "Exceptions" tab;

(32) Click on "Add Program…" button;

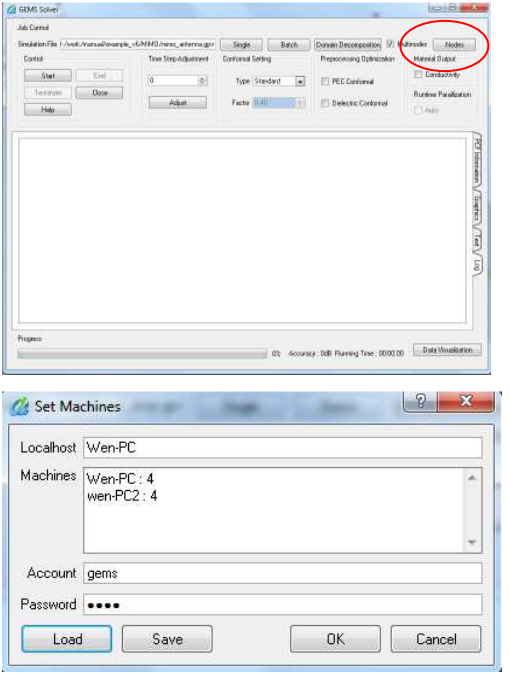
(33) Click on "Browse" button;

(34) Select "C:\programs files\GEMS\Bin\" folder.

(35) Select the "generator.exe" file;

(36) Select the "gems_solver.exe" file.

Run a test case, for example, you have two quad core processor computers. You can break the problem into 8 sub-domains and set the total number of cores to be 8.



You can run the job from either of the two computers following the same procedure. The current computer must be at the first place in the machine list.

1.3 Install GEMS Package on Linux Workstation

GEMS installation on a workstation with Windows system follows the exactly same procedure as that on a PC computer. For a Linux workstation, you need to

install the Linux operating system, GEMS WBI, and GEMS package on the workstation.

1.3.1 Prerequisite

You need to have the following software packages below you install GEMS on a Linux workstation:

(1) PuTTY, a free telnet/SSH client, can be downloaded from the following website:
"http://www.chiark.greenend.org.uk/~sgtatham/putty/download.html"

(2) WinSCP, a free SFTP, FTP and SCP client for Windows, can be downloaded from the following website
"http://winscp.net/eng/download.php"

(3) GEMS Linux Package, GEMSSimulator-XXX.tar.gz is provided by 2COMU.

(4) GEMS WBI Package, GEMSWBI.tar.gz is provided by 2COMU.

1.3.2 GEMS Package Installation

Follow the steps below to install GEMS package:

(1) Login as "root", transfer GEMS Linux Package, GEMSSimulator-XXX.tar.gz, to the Linux workstation. (WinSCP)

(2) Login as "root", type the following commands to unzip GEMS Linux Package to the folder "/opt". (PuTTY)

cd /opt [ENTER]

tar –xzf ~/GEMSSimulator-XXX.tar.gz [ENTER]

NOTE: "~" needs to be replaced by the absolute path if the GEMS Package is not in the root's home folder. "XXX" needs to be replaced by the version number of GEMS Package.

1.3.3 Parallel Runtime Environment Setup

GEMS cannot be run as "root". Login as user's account. (PuTTY). Follow the steps below to setup the MPI parallel runtime environment:

(1) Create authorization key for SSH.

ssh-keygen –t rsa [ENTER]

cd ~/.ssh [ENTER]

cp id_rsa.pub authorized_keys [ENTER]

(2) Modify the user's startup profile "~/.bashrc". Add the following lines at the end of the file.

```
# GEMS
if [ -f /opt/GEMS/script/envs.sh ]; then
    . /opt/GEMS/script/envs.sh
fi
```

NOTE: Important variables in "envs.sh": (i) OMP_NUM_THREADS=6， number of cores per slot; (ii) MPI_NUM_SLOTS=8, number of slots per node; (iii) Login again to make the change effective.

1.3.4 HTTP Server

(1) Login as "root", transfer HTTP Server and HTTP.tar.gz to the workstation. (WinSCP)

(2) Login as "root", type the following commands to unzip HTTP Server to the folder "/opt". (PuTTY)

cd /opt [ENTER]
tar –xzf ~/HTTP.tar.gz [ENTER]

NOTE: "~" needs to be replaced by the absolute path if the HTTP Server is not in the root's home folder. HTTP Server contains two sub folders, "httpd" and "php".

1.3.5 QUEUE Management

(1) Login as "root", transfer TORQUE, torque-package-server-linux-x86_64.sh, torque-package-clients-linux-x86_64.sh, torque-package-mom-linux-x86_64.sh, torque-package-doc-linux-x86_64.sh, torque-package-startup-linux.tar.gz and torque-package-devel-linux-x86_64.sh, to the Linux workstation. (WinSCP)

(2) Login as "root", type the following commands to install QUEUE Management. (PuTTY)

./ torque-package-server-linux-x86_64.sh --install [ENTER]
./ torque-package-clients-linux-x86_64.sh --install [ENTER]
./ torque-package-mom-linux-x86_64.sh --install [ENTER]
./ torque-package-doc-linux-x86_64.sh –instal l [ENTER]
./ torque-package-devel-linux-x86_64.sh --install [ENTER]
(optional)
cd /etc/init.d [ENTER]
tar –xzf ~/ torque-package-startup-linux.tar.gz [ENTER]

NOTE: "~" needs to be replaced by the absolute path if the torque-package-startup-linux.tar.gz is not in the root's home folder.

(3) To add "/opt/torque/bin" to searching path, open and append the following three lines to "/etc/profile":

#TORQUE

PATH=$PATH:/opt/torque/bin
export PATH

NOTE: Login again to make the change effective.

(4) Edit "/var/spool/torque/server_name" and replace the current value "wsa" with the workstation's hostname.

(5) Add QUEUE Management services with following commands.

chkconfig --add pbs_server [ENTER]
chkconfig --add pbs_mom [ENTER]
chkconfig --add pbs_sched [ENTER]
chkconfig pbs_server on [ENTER]
chkconfig pbs_mom on [ENTER]
chkconfig pbs_sched on [ENTER]

(6) Start QUEUE Management services with following commands.

service pbs_server start [ENTER]
service pbs_mom start [ENTER]
service pbs_sched start [ENTER]

(7) Configure QUEUE Management with following commands.

qmgr –c "create queue default" [ENTER]
qmgr –c "set queue default queue_type = Execution" [ENTER]
qmgr –c "set queue default enabled = True" [ENTER]
qmgr –c "set queue default started = True" [ENTER]
qmgr –c "set server scheduling = True" [ENTER]
qmgr –c "set server acl_host_enable = False" [ENTER]
qmgr –c "set server default_queue = default" [ENTER]
qmgr –c "set server query_other_jobs = True" [ENTER]

qmgr –c "create node XXXX np=4" [ENTER]

NOTE: "XXXX" needs to be replaced by the hostname set in "/var/spool/torque/server_name", "np=4" specify the number of cores in the workstation, for instance, 4.

1.3.6 Authorization Module of WBI

(1) Login as "root", transfer Authorization Module of WBI and WBI-Auth.tar.gz to the Linux workstation. (WinSCP)

(2) Login as "root", compile and install Authorization Module of WBI with following commands.

tar –xzf WBI-Auth.tar.gz [ENTER]
cd WBI-Auth [ENTER]
make [ENTER]
make install [ENTER]

1.3.7 WBI Configuration and Running

(1) Login as "root", edit "/opt/httpd/htdocs/gems/global/config.php".

In line 143, ***define("COMPUTING_NODES", 1)***, the number of nodes in the cluster, "1" for single machine
In line 144, ***define("COMPUTING_CORE_PER_NODE", 48)***, the number of cores in one node, for instance, 48

In line 145, ***define("COMPUTING_CORE_PER_SLOT",6)***, the number of cores per slot, for instance, 6

(2) Login as "root", execute the following command to start WBI service.

/opt/httpd/bin/apachectl start [ENTER]

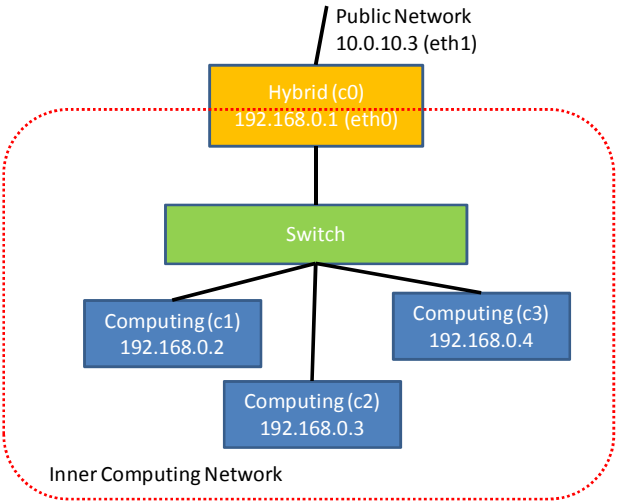
1.4 Install GEMS Package on Unmanaged Workstation Cluster

It is an efficient way to build an unmanaged workstation cluster if the number of computing nodes is less than four. All nodes in this type of cluster are involved in simulation and the cluster has no head node. We can use one of the computing nodes to be a hybrid node and the rest of them are used as the computing nodes.

The basic installation for the unmanaged workstation cluster is similar to the installation on a single workstation. The additional configurations include file based DNS, dsh (distributed shell), rsync (file synchronization) and NFS (network file system) setup.

VLAN Setup

Use the VLAN technique to split the inner computing network from the public network to avoid the unnecessary transmission collision and get the high parallel performance. It requires that the hybrid node have at least two network sockets. One is for the public network and the other is for the inner computing network.



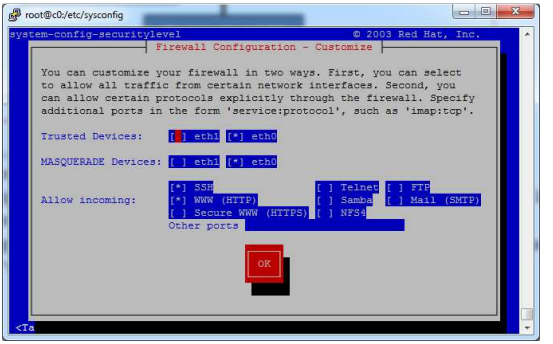
The hybrid node plays a gateway role and needs the following configurations.

(1) Enable ipv4 forward in the Linux kernel

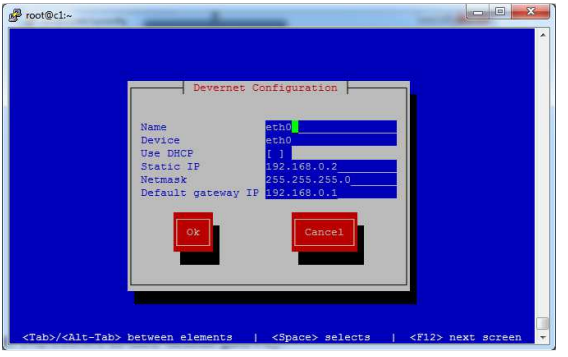
Add "net.ipv4.ip_forward = 1" in the file "/etc/sysctl.conf" to enable ipv4 forward in the kernel.

(2) Modify the firewall rule to do "IP MASQUERADE" on device eth0

Execute "setup" in the command line, and select "Firewall configuration", and select "Customize" button, make "eth0" as your trusted device and MASQUERADE device as well.



Set 192.168.0.1 as their default gateways on the other nodes.



File Based DNS

The following lines are appended to "/etc/hosts" in the hybrid and computing nodes to let all the nodes know each other by the given names

192.168.0.1 c0
192.168.0.2 c1
192.168.0.3 c2
192.168.0.4 c3

"c0/192.168.0.1" is the hybrid node.

DSH Setup

The dsh installation is only installed on the hybrid node and is a quite simple procedure for Linux distribution of CentOS and RedHat. We can start to install it by the "yum" command as follows:

yum install dsh

You need to create a new file "/etc/machines.list" that includes the following lines:

c0
c1
c2
c3

RSYNC Setup

We can install "rsync" on the hybrid and computing nodes via the following command:

yum install rsync

NFS Setup

We need to install NFS on the computing nodes and hybrid node separately following the steps given below:

1. Check the installation of NFS service.

The NFS might be already installed during the OS installation. You can try the following command to check

rpm -qa | grep nfs-utils

If you can find "nfs-utils-lib" and "nfs-utils" are installed in the current system, you can ignore the next step.

2. Install NFS service if it hasn't been installed yet.

The NFS can be installed via "yum" command as well

yum install nfs-utils

Follow the steps below to install NFS on the hybrid node:

3. Make sure NFS service is started automatically on boot.

chkconfig nfs on

4. Export (Share) the "/home" folder.

The following line is appended to "/etc/exports", which makes all the machines to share the entire "/home" directory in the IP address range "192.168.0.0~192.168.0.255".

/home 192.168.0.0/255.255.255.0(rw,async,no_root_squash)

You need to change the configuration "192.168.0.0/255.255.255.0" based on your own IP addresses for all computing nodes.

5. Restart NFS service to make the configuration effective.

service nfs restart

Follow the steps below to install NFS on the computing nodes:

6. Rename the current "/home" folder and create an empty new "/home" folder

rename /home /home_OLD
mkdir /home

7. Mount (Load) the "/home" folder exported by the hybrid node

   The following line is appended to "/etc/fstab", which will mount the "/home" folder on the hybrid node to the local "/home" folder.

c0:/home /home nfs
rw,soft,rsize=8192,wsize=8192 0 0

"c0" is the name of hybrid node.

8. Reboot the other nodes to make the changed effective.

Parallel environment setting is only carried on the hybrid node. GEMS WBI is only installed on the hybrid node. Queue Management Setup is only installed on both the hybrid and computing nodes, see the Section 1.3.5 for details.

You only need to perform the following four installations in the computing nodes

torque-package-clients-linux-x86_64.sh
torque-package-mom-linux-x86_64.sh
torque-package-doc-linux-x86_64.sh
torque-package-devel-linux-x86_64.sh

The most important is the startup scripts you put into "/etc/init.d". The only script you need put there is "pbs_mom". Then you only need to execute the following three commands on the other nodes

chkconfig --add pbs_mom
chkconfig pbs_mom on
service pbs_mom start

You also need add the other computing nodes during the queue management configuration (1.3.5) on the hybrid node with the command "qmgr" as follows:

…

The original configuration for the single workstation, including "c0"

qmgr –c "create node c0 np=4"

…

qmgr –c "create node c1 np=4"
qmgr –c "create node c2 np=4"
qmgr –c "create node c3 np=4"

Follow the steps below to open a user account:

To open the account on the unmanaged workstation cluster requires a special procedure. We can simply the procedure by the given script, "sync-users.sh", as follows. We can put the script into a proper folder in the current PATH. For instance, you can put it into "/root/bin", the "bin" folder of root account on the hybrid node.

```
#!/bin/sh
machines="c1 c2 c3"
for f in $machines; do
        rsync -v -t /etc/passwd root@$f:/etc/passwd
        rsync -v -t /etc/group root@$f:/etc/group
        rsync -v -t /etc/shadow root@$f:/etc/shadow
done
```

The line, machines="c1 c2 c3" need to be modified by your own names of the other computing nodes.

With this script, what we need to do are: (i) open an account on the hybrid node (ii) execute the script on the hybrid node.

Since it is a cluster now, we can significantly simply GEMS installation by using the DSH command. We first need to upload the GEMSSimulator-XXX.tar.gz into the "/home" folder on the hybrid node. Create a "temp" sub-folder in the home folder to keep the GEMS package. Then we can execute the following command to perform GEMS installation:

dsh –a tar –xzf /home/temp/GEMSSimulator-XXX.tar.gz –C /opt/GEMS

1.5 Install GEMS Package on High Performance Cluster

For a typical high performance cluster, the Racks cluster management is required. In this section, we describe a step-by-step guideline to install Racks and GEMS Linux version on your cluster. The GEMS installation is required by the cluster administrator.

1.5.1 Install Racks

If your cluster has been installed with Racks, you can skip this section. If your cluster has not installed Racks yet, you need to install Racks first following the steps given below:

(1) Getting Started

(i) Supported Hardware

Rocks is a cluster management solver that includes the CentOS Linux operating system. It only supports x86, x86_64 and IA-64 architectures

(a) Processors

- x86 (Intel ia32, AMD Athlon, etc.)
- x86_64 (AMD Opteron and Intel EM64T)
- IA-64 (Intel Itanium)

(b) Networks

- Ethernet
- Myrinet

- Infiniband

(ii) Minimum Hardware Requirements

(a) Frontend Node

- Disk Capacity: 20 GB or more
- Memory Capacity: 1 GB or more
- Ethernet: 2 Ethernet Card

(b) Compute Node

- Disk Capacity: 20 GB or more
- Memory Capacity: 1 GB or more
- Ethernet: 1 Ethernet Card

(2) Software Requirements

(i) The following packages are needed by GEMS

- Rocks 5.0
- Intel development Roll
- Torque Roll
- Service pack Roll

(ii) Rocks 5.0 can be downloaded from the following website

http://www.rocksclusters.org/wordpress/?page_id=82
You need to select x86_64 CDs or jumbo (DVD)

(iii) Intel development Roll can be downloaded from the following website

http://www.clustercorp.com/rolls/
You need register to download the roll. Make sure to download 64bit package。

(iv) Torque roll can be downloaded from the following website:

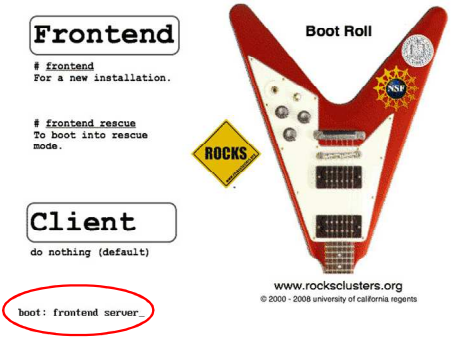
ftp://ftp.uit.no/pub/linux/rocks/torque-roll/5.0.0/torque-5.0.0-1.x86_64.disk1.iso
Burn the downloaded software onto CD or DVD, and label them for the later use.

(v) Install and Configure Your Frontend

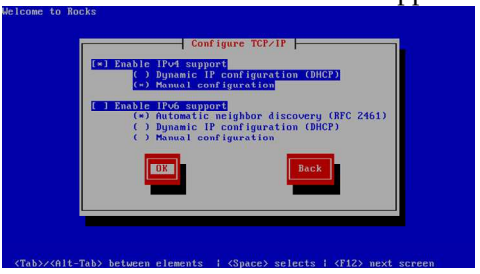
(a) Power on the front node computer.
(b) Change the BIOS boot sequence, make CD ROM your first boot device.
(c) Insert the DVD into the CD-ROM of the front node computer.
(d) After the booted from CD, you will see the following screen.

(e) Type "frontend server" after "boot:"

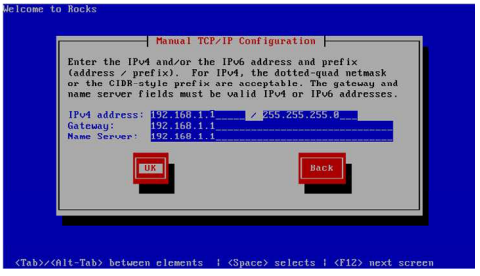
If you do not input the "frontend server" in several seconds, setup program will treat this computer as a compute node.



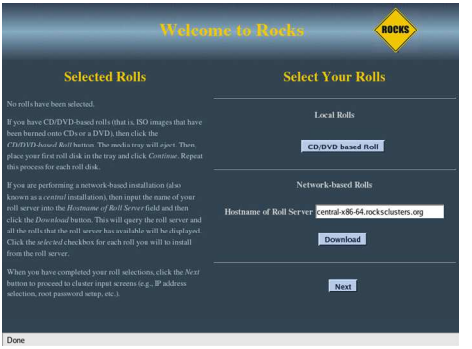
(f) It will take a while for the setup program to scan the hardware and then you will see the following screen. You need to select "Enable IPv4 support" and select "Manual configuration". You can unselect "Enable IPv6 support"



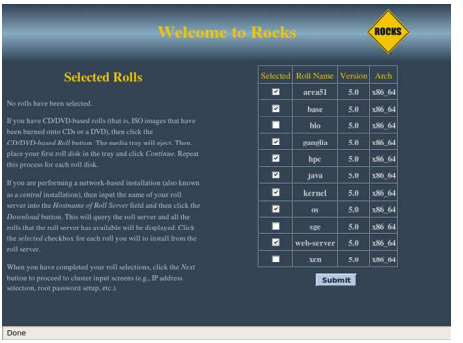
(g) Input the IP address for the ether-net card that connects the cluster to the network. You can get the information from your network administrator.



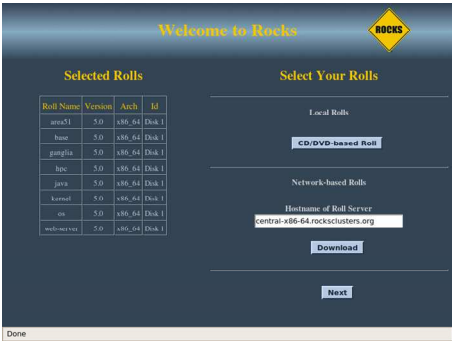
(h) You will see the following screen. Click on "CD/DVD-based Roll."



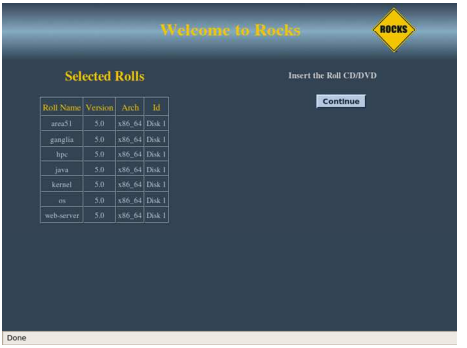
(i) In the following screen, select the necessary rolls as bellow, and then click on the "Submit" button.



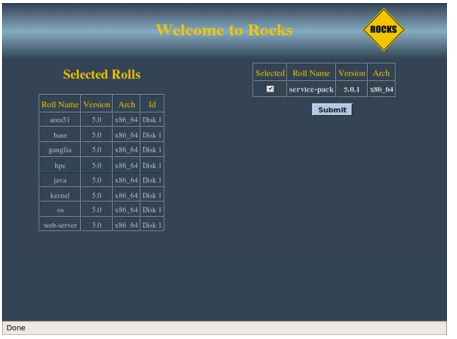
(j) You will see the screen with the Rolls you selected. Click on
(k) the "CD/DVD-based Roll" button again.



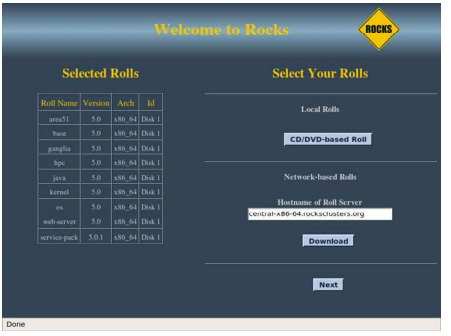
(l) CD will automatically be ejected. You will see the following screen, and insert "Rocks service pack roll" CD, and click on the "continue" button.



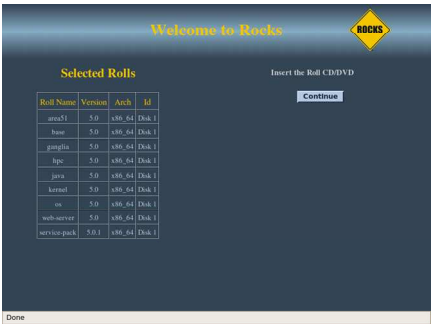
(m) Select "service pack" Roll, and click on "Submit" button.



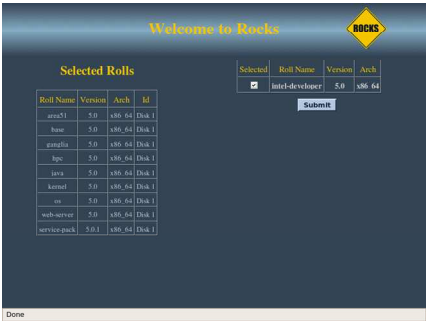
(n) You will see the screen with the Rolls you selected. Click on "CD/DVD-based Roll" again.



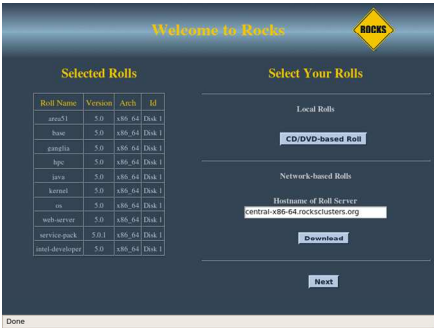
(o) The CD will be ejected. You will see the following screen. Insert your "Intel Developer Roll" CD, and click on "continue" button.



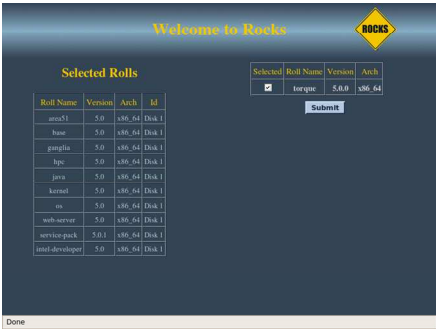
(p) Select "Intel Developer" Roll, and Click on "Submit" button.



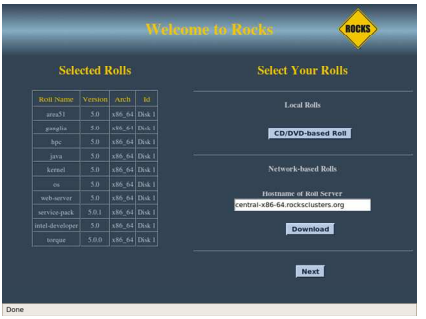
(q) You will see the screen with the Rolls you selected. Click on "CD/DVD-based Roll" again.



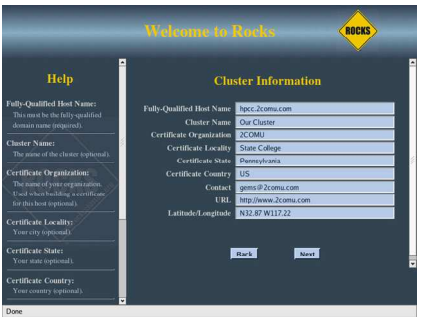
(r) Insert Torque Roll CD, and select "torque" Roll, and Click on "Submit" button.



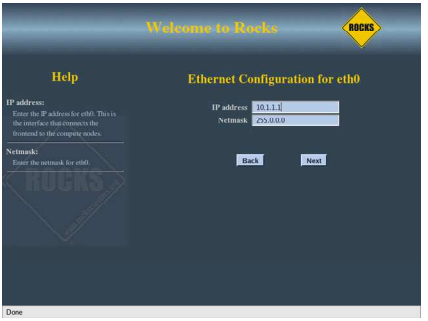
(s) After you select all the Rolls, you will see the following screen. Click on "Next" button.



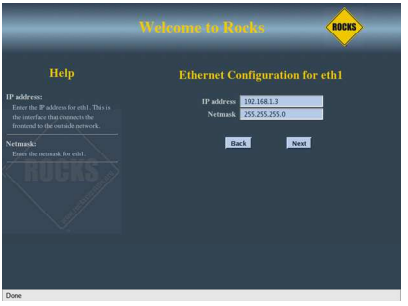
(t) Input the cluster information and then click on the "Next" button.



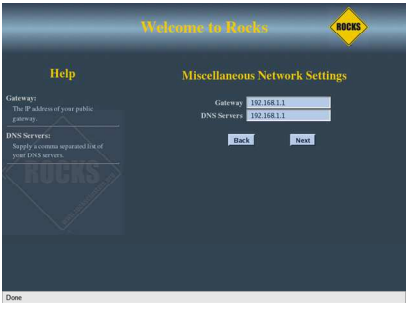
(u) You are recommended to use the default setting to setup the internal network IP address. Click on the "Next" button.



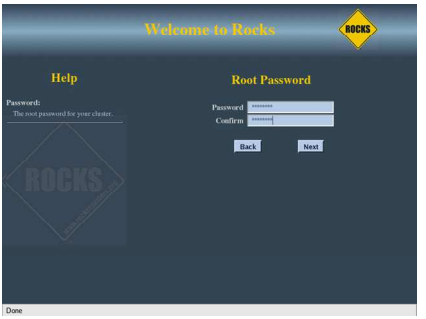
(v) Input network information you got from network administrator. This network card is the interface that connects the frontend node to the outside network. Click on the "Next" button.



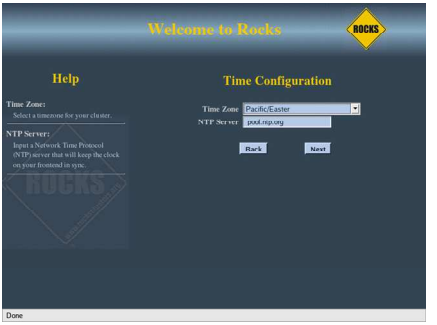
(w) Input Gateway and DNS server information you get from the network administrator. Click on "Next" button.



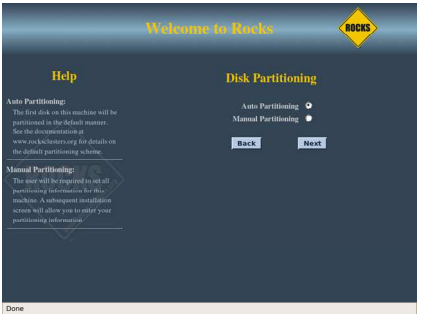
(x) Input your root password. To make the network safe, the password should include letters, numbers and at least one uppercase letter.



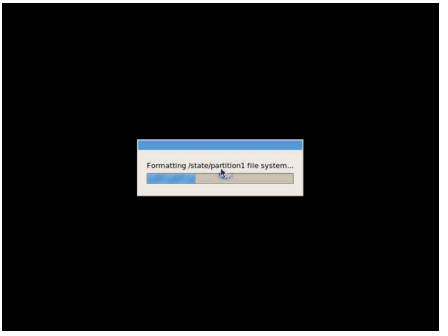
(y) Select your time zone and Network Time Protocol server.



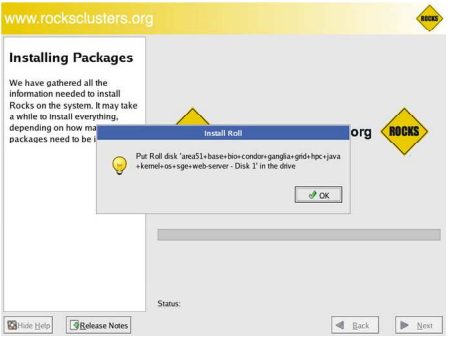
(z) Disk Partitioning screen allows you to partition your hard disk. There are two options, namely, Auto Partitioning and Manual Partitioning. We recommend you select "Auto Partitioning".



(aa) Installation program will partition and format the hard disk automatically.

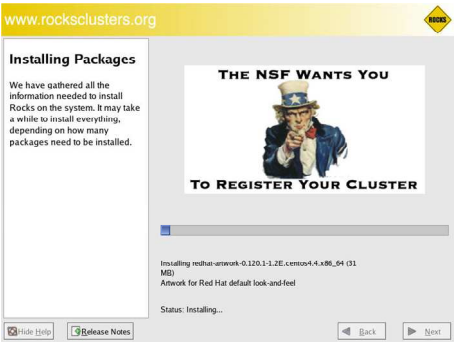


(bb) You will be prompted to insert the Rocks Roll DVD. Insert the DVD and click on the "Ok" button.

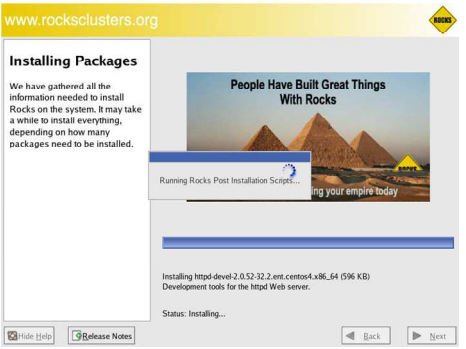


(cc) Insert "Intel developer roll", "service-pack roll" and "torque roll" disk, respectively, as the prompted message appears and then click on the "Ok" button.

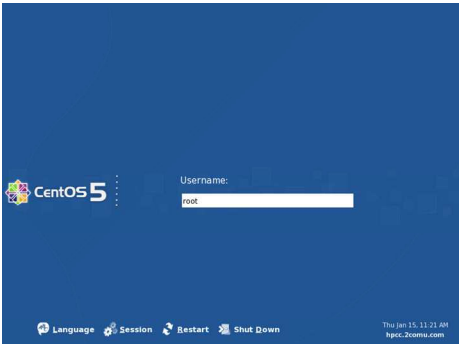
(dd) Installation process will begin.

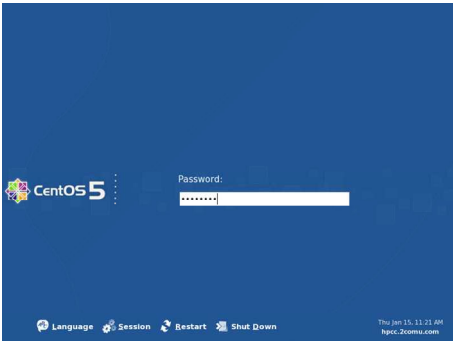


(ee) After Post Installation Scripts finished, the computer will restart.

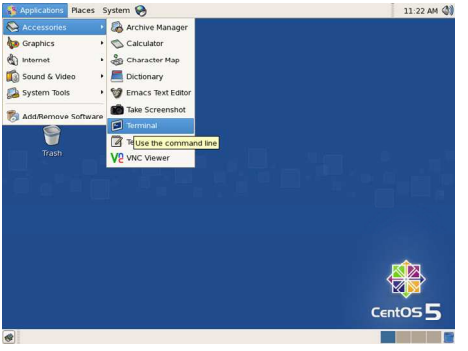


(ff) When the computer first start, login as root.

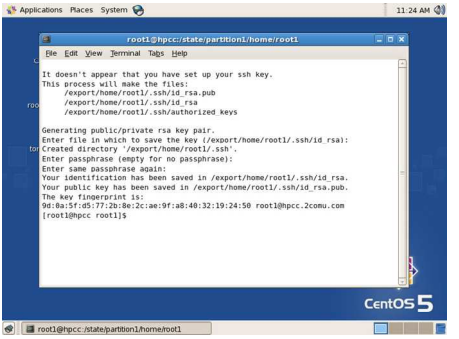




(gg) Start X Windows Terminal.



(hh) Press Enter key to keep all default settings.



(3) Install pam development package 32bit version.

(i) Login frontend as root.

(ii) Start X Windows Terminal

(iii) Go to pam development package folder using the command

(iv) cd /export/home/install/rolls/os/5.0/x86_64/RedHat/RPMS

(v) Install pam package using the command

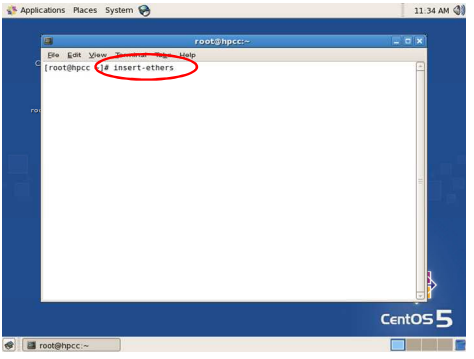
(vi) rpm -ihv pam-devel-0.99.6.2-3.26.el5.i386.rpm

(4) Install compute node

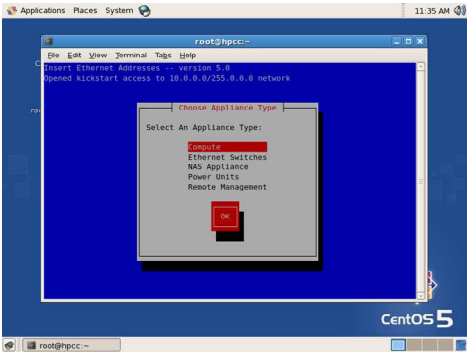
(i) Login frontend as root.

(ii) Start X Windows Terminal

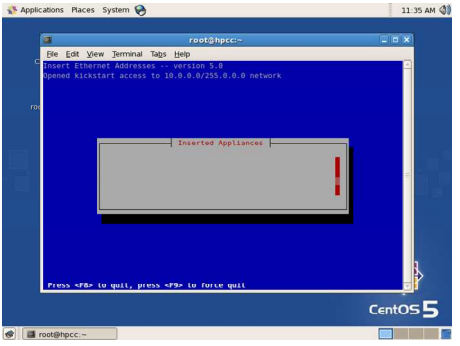
(iii) Type "insert-ethers"



(iv) the screen like this. Select "Compute" and press "Ok"

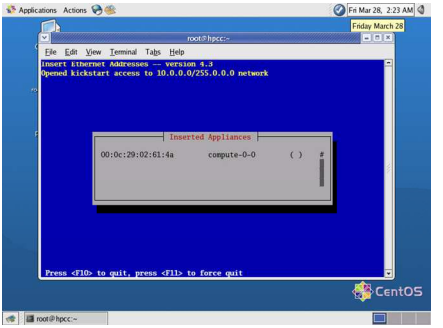


(v) You will see the screen like bellow.

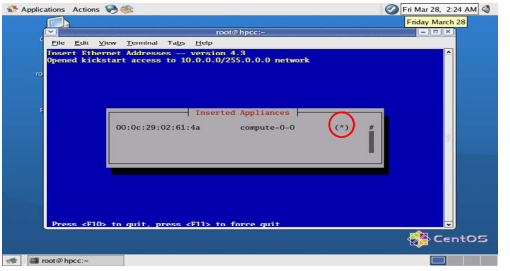


(vi) Power on compute node. If both the compute node and the switch support PXE, boot compute node with PXE, otherwise, boot computer node with CD-ROM.

(vii) The frontend will capture compute node network information and assign the node with a unique name.



(viii) You will see a "*" if the compute node is successfully inserted.



(ix) Repeat the above steps to insert other compute node.

(5) User management

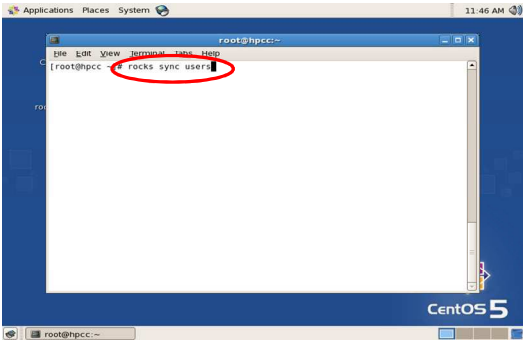
(1) Login frontend as root

(2) Use "useradd –n username" command to a user

(3) Modify the user password using the command

passwd username

(6) Type "rocks sync users" for new added users to synchronize the user information in the cluster.



(7) Setup user default security information using the command (It is very important to do this , otherwise web job submitation maybe fail.)

su uername

1.5.2 Install GEMS Package

If your cluster has been installed with Racks, you can follow the steps below to install GEMS software:

(1) Upload GEMSSimulator-<Version>.tar.gz to the cluster

(2) Unpack GEMSSimulator-<Version>.tar.gz with the command

"tar -xzf GEMSSimulator-<Version>.tar.gz"

(3) Change the current login to "root" with the command "su -"

(4) Move the unpacked folder into /share/apps with the command

"mv GEMS /share/apps/GEMS"

(5) Copy the license file "gems.lic" to the folder "/share/apps/GEMS/bin64"

"cp gems.lic /share/apps/GEMS/bin64/gems.lic"

1.5.3 Install GEMS WBI

GEMS WBI allows the users to access to the cluster through web browser Firefox or IE. Follow the steps below to install GEMS WBI on your cluster.

(1) WBI Installation:

(i) Login as root

(ii) Upload WBI-package.tar.gz

(iii) Unpack WBI-package.tar.gz using the command

```
tar -xzf WBI-package.tar.gz
```

(iv) Enter the folder WBI using the command

```
cd WBI-package.
```

(v) Build WBI webpages

```
cd gems
./install.sh
```

(vi) Build WBI authorization module

```
cd auth
make
make install
```

(vii) Install ftp server

```
cd vsftpd
./vsftpd-install.sh
```

(2) WBI Post-Installation:

(i) Modify the maui configuration and change the policy for job node match.
Open the file "/opt/maui/maui.cfg" and add a new line in "Node Allocation" section. (about at line 43)

```
ENABLEMULTIREQJOBS  TRUE
```

Restart maui service using the command "service maui restart".

(ii) Modify the default maximum size of the upload files to be a proper value.

Open the file "/etc/php.ini" and find "upload_max_filesize" in "File Uploads" section(at line 582). For example,

"upload_max_filesize = 128M" will set the max size to 128MB.

Find "post_max_size"(at line 477) and make sure its value is equal or larger than the value of "upload_max_filesize". For example,

"post_max_size = 128M"

Find "memory_limit" at the end of this file(at line 1222) and give a value no less than the value of "post_max_size". For example,

memory_limit = 256M

(iii) Open /var/www/html/gems/global/ config.inc using command

vi /var/www/html/gems/global/ config.inc
Modify define("COMPUTING_NODES",x); x is the computer node number
Modify define("COMPUTING_CORE_PER_NODE",y); y is the number of cores in each node

Modify define("CPU_MAX_NUMBER", z); z should be x * y

(iv) Restart the web service using the command

"service httpd restart"

to make the modifications effective.

1.5.4 Comments

(1) The root user cannot submit job. The root opens regular user account and the regular user can submit jobs.

(2) If a job is submitted, a job ID is assigned, otherwise, the submission is failed, for example, the message like "your job has submitted".

To fix the problem by yourself, follow the steps below:

Go to the GEMS project folder in the terminal window, type

"qsub *.pbs",

For "qsub: Unknown queue MSG=cannot locate queue", switch from the current user to root, type

"qmgr < /opt/torque/pbs.default".

Switch root to the current, submit the job again.

(3) Ref: **Rocks Cluster Distribution: Users Guide**

http://www.rocksclusters.org/rocks-documentation/5.0/index.html

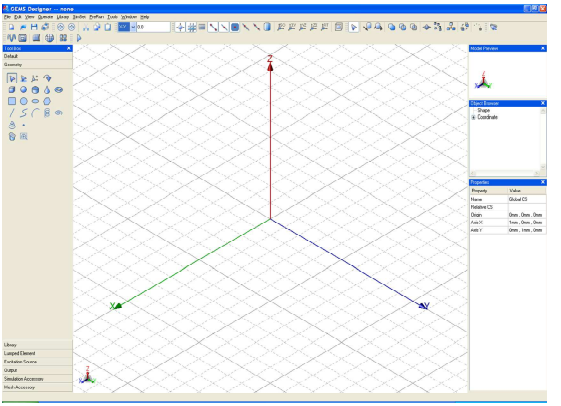
Chapter 2 Introduction to Designer

GEMS provides both Windows and Linux modeling graphical interfaces, which have the identical appearance in Windows and Linux platforms. To run the GEMS projects, you can generate the project files in either Windows or Linux platforms. In this guide, all the screen shots are captured from Windows platform.

2.1 GEMS Designer Graphical Interface

The GEMS designer is used to generate the project model including geometry, mesh, material information, boundary condition, and information of excitation source and output parameters. Selecting the **designer** option 

from the GEMS folder in the **START** program will open the designer window.



There are 5 major parts in the **designer** window: the toolbar and the file menu; the **Tool Box**; the graphical display region, the **Model Preview** region; the **Object Browser** box, and, the **properties** box.

2.2 Edit and View Status

The objects in the picture region have two statuses: (i) edit status; (ii) view status. The  button, which serves as a switch of two statuses, is located in the toolbar and the tool box.

GEMS allows you to select a surface of object by the surface selection button  , which is located in the tool box.

You can select one object by moving the mouse icon to the object and press the left mouse button, or, select the object entry in the **Object Browser** box. Select the first object, you can press and hold the **Ctrl** key on the keyboard to select more than one object. You can also select all the objects by pressing and holding "Ctrl + a" keys.

2.3 Basic Modeling Procedure

The basic procedures to draw an object in the graphical interface:

(1) Move the mouse icon to the picture region.

(2) Select the desired position and press the left mouse button to confirm the selected point.

(3) Release the mouse button.

(4) Move the mouse icon to the next position and press the left mouse button when the mouse icon is the desired place.

(5) Release the mouse button and move the next point until the last point.

(6) GEMS allows you to snap the mouse icon to the background grids by pushing the  button down, or move the icon to any place in the picture region by pulling the button up.

(7) GEMS also allows you to snap the mouse icon to the different position of line segment, or surface center.

(8) GEMS provides both the absolute coordinate system and relative coordinate system.

(9) You can draw a problem structure using the functions provided by GEMS, or, you can import a CAD file to GEMS interface.

(10) The domain decomposition module in GEMS allows you to use a single PC or a cluster efficiently process the material and conformal mesh distributions on either Windows or Linux platform.

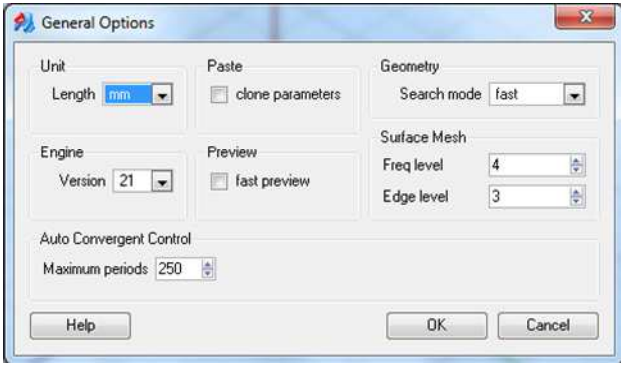
(11) GEMS allows you to map a planar design to the surface of a 3-D structure.

Chapter 3 Introduce Buttons

In this section, we introduce the major buttons and options in the **GEMS** designer. Each button in the toolbar has the corresponding counterpart in the file menu. We will describe each button in the toolbar and representative options in the file menu.

3.1 Project Unit

The default length unit is located in the **Set Default** option in the **Tools** menu. Any length parameter in the project without the unit specification will be taken to be the default unit. This default unit is defined at the beginning when a project is created, and it cannot be modified though you can use different units for different input variables during the project design. You need to specify a time unit for the time variable in the excitation delay specification; otherwise, GEMS will use the default unit (nanosecond (*ns*)). No matter what unit you choose, GEMS will convert all the units into *meter*, *second* and *kilogram* in the GEMS simulation.



Unit->Length： default length unit.

Paste->Clone parameters: Checked: the parametric variables in the pasted object will have a different names.
Unchecked: the parametric variables in the pasted object will have the same names.

Geometry->Search mode: Using the different ways to search the key points during the mesh generation.

Fast：more efficient but may not find some key points inside the complex geometry;
Accurate：more accurate but is slow.

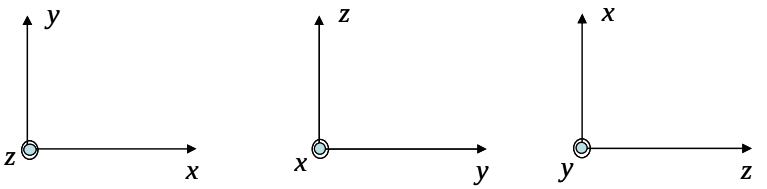
Engine->Version： number of ACIS version

Preview->fast preview： The rotated or moved geometry only shows the outside border without details if the box is checked.

Surface Mesh->Frequency level: The number of the longest length of triangle per wavelength for generating the triangular meshes.
Surface Mesh->Edge level: The partition number of the model edge for the generating the triangular meshes.
Two options above are designed for the conformal surface current output.

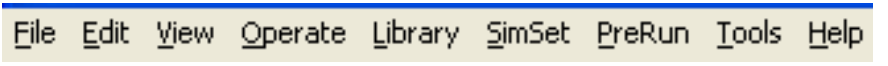
Auto Convergent Control->Maximum periods: The maximum number of time periods is allowed for GEMS simulation.

GEMS follows the right hand rule throughout; specifically, it uses x (horizontal or width); y (vertical or height)) in the x-y plane (y (horizontal or width); and z (vertical or height)) in the y-z plane and (z (horizontal or width), x (vertical or height)) in the z-x plane.



3.2 File Menu

The options in the file menu, which are located at top of the **designer** window, represent most of the major features of the GEMS designer. Most of their names and classifications are based on the common sense, and then, you will be able to recognize their meanings from their names.



3.2 Tool Bar

The toolbar includes the most frequently used functions in the modeling, and can be divided into the following groups.

File operation buttons: 
Description: Create a new project; open a project; save a project; and, print the picture area.

Object operation buttons: 
Description: Object undo; object redo; object or option delete; object or option copy; and, object or option paste.

Drawing reference: 

Description: Drawing plane and its height.

Display option buttons:

Description: Grid snap switch; background grid switch; mesh switch; snap to ends of a line; snap to the center of a line; snap to the center of surface; snap to one-third of a line; snap to quarter of a line; object transparence switch; reset to ISO; x-y plane view, y-z plane view and z-x plane view; fit to the display area automatically; options (simulation domain, Huygens' box, and plane wave source) view.

Object operation buttons:

Description: Edit or display switch; object move; object rotation; object mirror; object unite; object subtraction; object intersection; formula transform; object hardclone; object group; object order adjustment; and, object sweep.

Simulation buttons: 

Description: Excitation pulse setup; domain and boundary condition setup; mesh design; excitation or/and output mode extraction; parallel processing design; and, project simulation file generation.

3.3 Tool Box

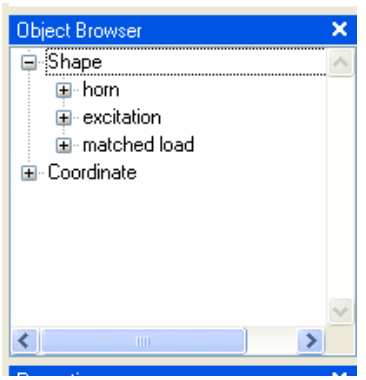
GEMS **Tool Box** integrates the basic geometrical types of objects and their associated properties. You can draw an object in the picture region and then assign a property to this object. There is an icon  in the each box, which is used to void the assigned property.

3.4 Project Preview Window

GEMS provides a preview sub-window in the **designer** window in order to view the project configuration during the project design. You can move the mouse icon into this sub-window to view the global configuration by rolling the middle mouse button.

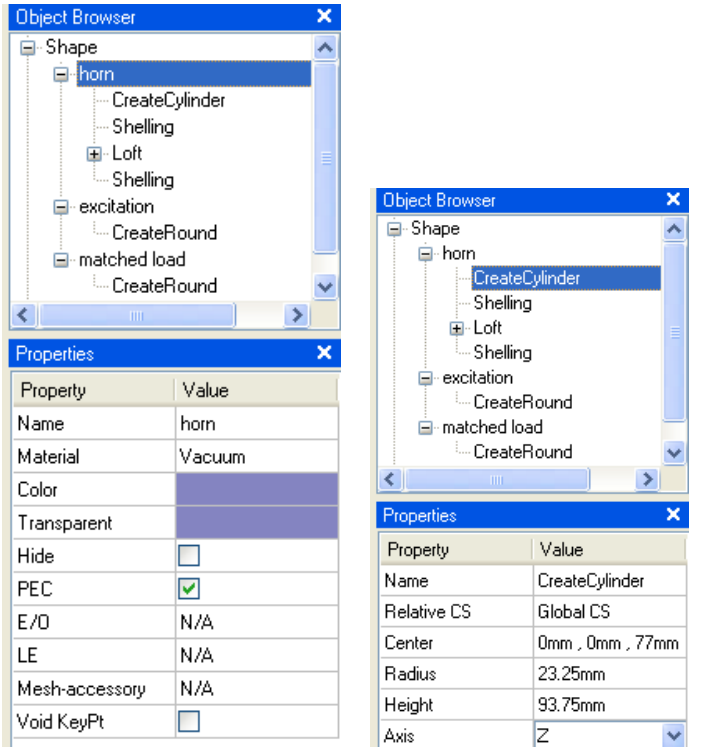
3.5 Object Browser Box

Each object in the picture region will have a corresponding entrance in the **Shape** tree in the **Object Browser** box. The object appearance and properties can be specified or modified through the options in the **Properties** box, which will appear by selecting and clicking on this object. Additionally, the operation history associated this object will also be listed under this object entrance.



3.6 Property Box

The properties associated with the option in the **Object Browser** box will be listed in the **Properties** box. These parameters in the **Properties** box can be modified or can be assigned a new parameter by clicking on it.



Two property boxes will appear by selecting and clicking the corresponding entrance in the **Shape** tree. If the object material type is the perfect electric conductor (PEC), you can check the **PEC** box, which has a higher priority than the material specified in the **Material**. If the **N/A** is in the **Material** box and the **PEC** box is unchecked, this object will be ignored in the current project. You can click on the box in the **Material** box to select the object material.

If the object is associated with an excitation source, you need to select this object first, and then select the desired parameters in the **Excitation Source** box of **Tool Box**. Specify the desired excitation parameters in the pop-up window, the corresponding name will appear in the **E/O** box.

The **Fine-mesh** option allows you to use a fine mesh distribution inside the object (or region). Click on the **Fine-mesh** option, GEMS allows you to specify the subgridding information.

The density option is used to calculate the SAR distribution on a specified region. You can input a density value in the **Density** box.

The **LE** (lumped element) option is used to add a lumped element (resistor, capacitor or inductor) in the computational domain. Specify the desired lumped element parameters in the pop-up window, the corresponding name will appear in the **LE** box.

Next, we show you how to specify the geometry information in the **Properties** box. The **Relative CS** allows you to use a relative coordinate system to draw an object. Drawing a relative coordinate system is accomplished by clicking on the icon  in the **Tool Box**, following which you can draw an object referred to this relative coordinate system.

The types and number of the geometry parameters in the list depend on the object type. You will know what they refer to from their name in the list. However, the terminologies such as "width", "length" or "height" are defined based on the reference plane.

3.7 Picture Window

The picture window shows the project configuration such as objects, excitation sources and outputs. The background grids can be hidden by

clicking on the  button. Click on the  button to view the project model in the transparent style. The objects inside the picture window can be selected either by using the mouse or by clicking on the option inside the **Object Browser** box. The display window can be copied and pasted by clicking on the  button and then on the  button.

In order to select more than one object, you need to press and hold the **Ctrl** key and then press the left mouse button to select the option one by one. GEMS also provides a hot key that allows you to select all the options inside the picture region, namely, press and hold the **Ctrl** key and **a** key on the keyboard.

Chapter 4 Menu and Toolbar

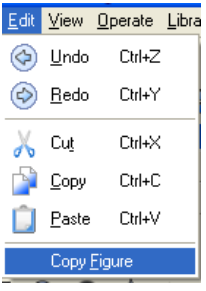
In this chapter, we introduce the menu and buttons in the toolbar in the GESM designer. The most frequently used functions are assigned to the special icons and placed in the toolbar. The complete functions are listed in the menu and putted in the different categories according to their meaning.

4.1 Menu

File menu

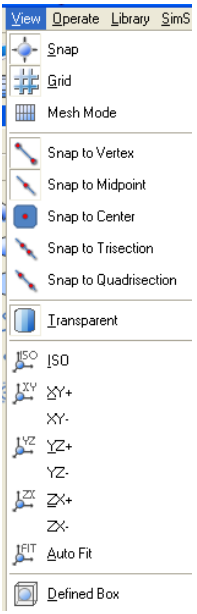
Create a new project
Open an existing project
Save current project
Save as current project
Import a CAD model
Export the model as a CAD file
Export the model in parts
Recent files
Print setup
Print preview
Print picture region
Exit GEMS designer

Edit menu



Undo an object operation
Redo am object operation
Delete an option
Copy an option to the clipboard
Paste an option from clipboard
Copy picture to clipboard

View menu



Snap the mouse icon to the background grid

Show the background grids

View the mesh distribution

Snap the mouse icon to ends of line segment

Snap the mouse icon to the middle point of a line segment

Snap the mouse icon to surface center

Snap the mouse icon to one-third of a line segment

Snap the mouse icon to quarter of a line segment

View object model in transparent style

Reset the model view status to the default

View model along the +Z direction

View model along the -Z direction

View model along the +X direction

View model along the -X direction

View model along the +Y direction

View model along the -Y direction

Fit the model to the picture region

View relative position of simulation region, Huygens' surface and plane wave surface

Operate menu

Select object

Move object

Rotate object

Select object surface and create a thickness

Generate a surface from closed polygon

Unite objects to single element

Subtract objects from others

Get the intersection of two objects

Get a object from an analytic expression

Generate an array with the same features

Generate an array that are independent

Solidify a closed shell

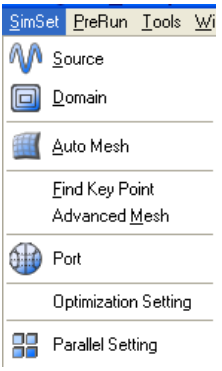
Generate a transition between two surfaces

Group objects as a single element

Adjust object order

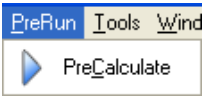
Sweep a line, surface or 3-D object to generate a complex structure

SimSet menu



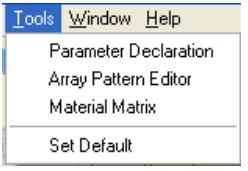
Design excitation source
Design domain and boundary
Generate adaptive mesh
Manually design mesh
Advanced mesh design
Extract mode pattern of excitation or output port
Optimization parameter setting
Parallel design

PreRun Menu



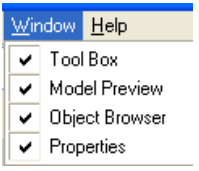
Project simulation file generation

Tools menu



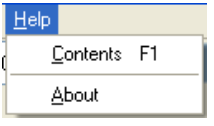
Parameter declaration for parametric design
Array design
Voxel data load for bio-electromagnetics
View and edit default options

Window menu



Display Tool Box
Display Model Preview window
Display Object Browser box
Display Properties box

Help menu



Display on-site help document
Show About information

4.2 Introduction to Icons in Tool Box

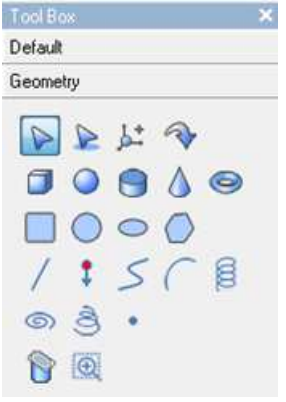
Default parameters:

Material: The default material when you draw an object in figure region.

Color: The default material when you draw an object in figure region.

Transparent: The default model is not transparent.

Grid size: The default grid size of figure region background is 0.2 mm.



Geometry:

 3-D cuboid (input parameters: length, width and height)

 3-D sphere (input parameters: coordinates of center point and radius)

 3-D cylinder (input parameters: coordinates of the reference point, radius and height)

 3-D corn (input parameters: coordinates of the reference point, radius and height, or )

 3-D torus (input parameters: major radius, minor radius and coordinates of the center point)

 2-D rectangle (input parameters: width and length)

 2-D circle (input parameters: coordinates of the center point and radius)

 2-D ellipse (input center point, vertex of major axis and ratio of minor to major radius)

 2-D polygon (input parameters: coordinates of each vertex)

 1-D line segment (input parameters: coordinates of two points)

 Plumb line (input parameters: coordinates of two points)

 Spline (input parameters: coordinates of each point on the line)

 1-D arc (input parameters: coordinates of center point, radius and angle range)

 3-D helix (input parameters in the dialogue window)

 3-D spiral (input parameters in the dialogue window)

 3-D spring (input parameters in the dialogue window)

 Point (input parameters: coordinates)

Several icons in the **Tool Box** are employed for the object operations:

 A switch between drawing and view statuses.

 Select object surface.

 Relative coordinate system.

 Import a CAD model.

 Cross section view of geometry.

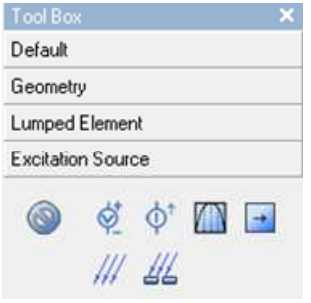
Lumped element

 Cancel property assignment

 Resistor

 Capacitor

 Inductor.



Excitation Source

 Cancel property assignment

 Voltage excitation

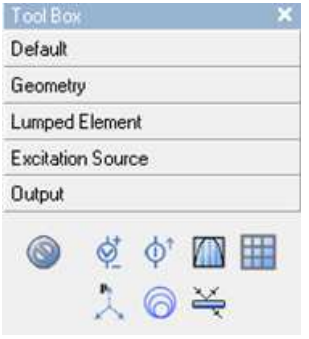
 Current excitation

 Mode excitation


Aperture excitation


Plane wave excitation


Periodic structure excitation



Output


Cancel property assignment


Voltage output


Current output


Mode output


Field distribution at point, on line and on surface


Poyting vector distribution and transmitted power through a specified surface


Far field output including far field, directivity, gain


Reflection and transmission coefficients of periodic structure

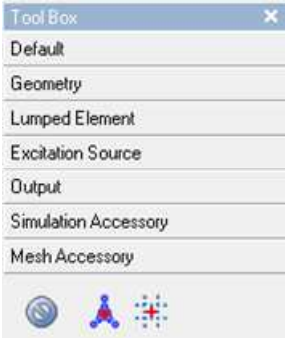
Simulation Accessory

 Cancel property assignment

 Lumped port (excite the open port)

 Wave port (excite the match port)

 Match load for open mode port



Mesh Accessory

 Cancel property assignment

 Key point for special mesh design

 Subgridding

Chapter 5 Objects and Operations

In this chapter we will introduce the procedure for drawing an object in the **Geometry** box. To begin with the object drawing, you need to select a proper background grid size in the **Default** box, and then go to the **Geometry** box to select the object type to be drawn. Select the drawing plane 

or 

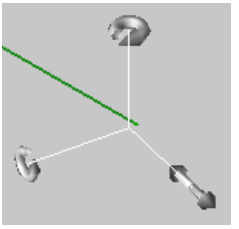
and the plane position 

. Click on the object icon and then move the mouse icon to the picture region, a black dot (default) will appear in the picture region. A pop-up text box at the bottom right will indicate the position of the mouse icon inside the picture region.

Clicking on the  button (press down) in the toolbar will only allow the mouse icon (black dot) to move on the grids of the background; however, releasing the  button (release up) will allow the mouse icon at any position in the picture area. If you select a proper grid size in the **Default** box, it is easier to select the desired position in the picture region. The background grid is shown by pressing down the  button and hidden by releasing up the  button.

During the modeling, the  button allows you to switch the statuses between the geometry edit and view. Pressing the  button down will allow you to select an object icon in the **Geometry** box and draw it in the picture region. Releasing the  button up will switch to the view status in which you can use the mouse to view the project configuration.

Although GEMS places the  button (geometry cutting view) in the **Tool Box**, its function is actually for the cutting view of the project geometry. In order to view the cross section of an object or inside structure, you can click on the  button, and three operating signs will appear in the picture region.



This sign will disappear when you release the button up. If the button is pressed down, the objects inside the picture region will be transparent; otherwise if the button is release up, the objects in front of it will block the objects behind it.

5.1 Single Object Drawing

Most of the object properties and attributes are summarized in the **Properties** box after an object has been successfully drawn. Therefore, you can modify the object properties and attributes from the **Properties** box. Next, we introduce how to draw a single object.

5.1.1 Cube

The basic steps are described below:

(1) Select the icon in the **Tool Box** and then move the mouse icon to the picture region.

(2) Select the drawing plane X-Y, X-Z or Y-Z.

(3) Choose a desired position (one of corners of a cube) in the selected plane, and then press the left mouse button to confirm the corner of cube.

(4) Move the mouse icon to draw a rectangle and then press the left mouse button to confirm the rectangle.

(5) Move the mouse icon in the vertical direction to draw a cube and then press the left mouse button to confirm the cube.

(6) The geometry parameters are listed in the **Property** box and can be adjusted if necessary.

5.1.2 Sphere

The basic steps to draw a sphere are described below:

(1) Select the icon  in the **Tool Box** and then move the mouse icon to the picture region.

(2) Select the drawing plane  ,  or  .

(3) Choose a desired position (center point) in the selected plane, and then press the left mouse button to confirm the center point.

(4) Move the mouse icon to draw a sphere, and then press the left mouse button to confirm the sphere.

(5) The geometry parameters are listed in the **Property** box and can be adjusted if necessary.

5.1.3 Cylinder

The basic steps to draw a cylinder are described below:

(1) Select the icon  in the **Tool Box** and then move the mouse icon to the picture region.

(2) Select the drawing plane  ,  or  .

(3) Choose the center point in the selected plane, and then press the left mouse button to confirm the center point.

(4) Move the mouse icon to draw a circle and press the left mouse button to confirm the circle.

(5) Move the mouse icon in the vertical direction to draw a cylinder and then press the left mouse button to confirm the cylinder.

(6) The geometry parameters are listed in the **Property** box and can be adjusted if necessary.

5.1.4 Cone

The basic steps to draw a cone are described below:

(1) Select the icon  in the **Tool Box** and then move the mouse icon to the picture region.

(2) Select the drawing plane  ,  or  .

(3) Choose the center point of base in the selected plane, and press the left mouse button to confirm the center point.

(4) Move the mouse icon to draw a circle and then press the left mouse button to confirm the circle.

(5) Move the mouse icon in the vertical direction to draw the height of cone and then press the left mouse button to confirm the height.

(6) Move the mouse icon to draw the circle and then press the left mouse button to confirm the cone.

(7) The geometry parameters are listed in the **Property** box and can be adjusted if necessary.

5.1.5 Torus

The basic steps to draw a torus are described below:

(1) Select the icon  in the **Tool Box** and then move the mouse icon to the picture region.

(2) Select the drawing plane  ,  or  .

(3) Choose a desired position (center point) in the selected plane, and press the left mouse button to confirm the center point.

(4) Move the mouse icon to draw a circle and press the left mouse button to confirm the circle of torus.

(5) Move the mouse icon to draw the cross section of torus and press the left mouse button to confirm the torus.

(7) The geometry parameters are listed in the **Property** box and can be adjusted if necessary.

5.1.6 Rectangle

The basic steps to follow to draw a rectangle are described below:

(1) Select the icon  in the **Tool Box** and then move the mouse icon to the picture region.

(2) Select the drawing plane  ,  or  .

(3) Choose a desired position (one of the rectangle corners) in the selected plane and then press the left mouse button to confirm the corner of rectangle.

(4) Move the mouse icon to draw a rectangle and press the left mouse button to confirm the rectangle.

(5) The geometry parameters are listed in the **Property** box and can be adjusted if necessary.

5.1.7 Circle

The basic steps to be followed to draw a circle are described below:

(1) Select the icon  in the **Tool Box** and then move the mouse icon to the picture region.

(2) Select the drawing plane  ,  or  .

(3) Choose a desired position (center point) in the selected plane and then press the left mouse button to confirm the center of circle.

(4) Move the mouse icon to draw a circle and then press the left mouse button to confirm the circle.

(5) The geometry parameters are listed in the **Property** box and can be adjusted if necessary.

5.1.8 Ellipse

The basic steps to draw an ellipse are described below:

(1) Select the icon  in the **Tool Box** and then move the mouse icon to the picture region.

(2) Select the drawing plane 

, or .

(3) Choose a desired position (center point) in the selected plane and then press the left mouse button to confirm the center point.

(4) Move the mouse icon to a circle and then press the left mouse button to confirm the circle.

(5) Move the mouse icon to draw the second axis and then press the left mouse button to confirm the ellipse.

(6) The geometry parameters are listed in the **Property** box and can be adjusted if necessary.

5.1.9 Polygon

The basic steps to draw a polygon are described below:

(1) Select the icon  in the **Tool Box** and then move the mouse icon to the picture region.

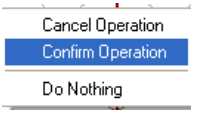
(2) Select the drawing plane 

, or .

(3) Choose the first vertex in the selected plane and then press the left mouse button to confirm the vertex.

(4) Repeat this procedure until the last vertex.

(5) Move the louse icon to the first vertex, and then press the left mouse button, or, press the right mouse button and select the **Confirm Operation** option in the Pop-up menu.



(7) The geometry parameters are listed in the **Property** box and can be adjusted if necessary.

5.1.10 Line

The basic steps to draw a line are described below:

(1) Select the icon  in the **Tool Box** and then move the mouse icon to the picture region.

(2) Select the drawing plane  ,  or  .

(3) Choose the initial point and press the left mouse button to confirm the first point.

(4) Move the mouse icon to draw a line, and then press the left mouse button, or, press the right mouse button and select the **Confirm Operation** option in the popup menu, to confirm the line.

(5) The geometry parameters are listed in the **Property** box and can be adjusted if necessary.

5.1.11 Spline

The basic steps to draw a spline are described below:

(1) Select the icon  in the **Tool Box** and then move the mouse icon to the picture region.

(2) Select the drawing plane  ,  or  .

(3) Choose the initial point and then press the left mouse button to confirm the initial point.

(4) Move the mouse icon to draw a curved line and then press the left mouse button to confirm the first segment.

(5) Repeat the procedure until the last point.

(6) Move the mouse icon to the last point, and then press the left mouse button, or, press the right mouse button and select the **Confirm Operation** option in the popup menu, to confirm the spline.

(7) The geometry parameters are listed in the **Property** box and can be adjusted if necessary.

5.1.12 Arc

The basic steps to be followed to draw an arc are described below:

(1) Select the  icon in the **Tool Box** and then move the mouse icon to the picture region.

(2) Select the drawing plane  ,  or  .

(3) Choose the center point in the selected plane and press the left mouse button to confirm the center point.

(4) Move the mouse icon to draw a line and then press the left mouse button to confirm the radius of arc.

(5) Move the mouse icon to draw an arc and then press the left mouse button, or press the right mouse button and select the **Confirm operation** in the popup menu, to confirm the arc.

(6) The geometry parameters are listed in the **Property** box and can be adjusted if necessary.

5.1.13 Helix

The detailed procedure is described below:

(1) Click on the icon in the **Tool Box**, move the mouse icon into the picture region, and select the center point of helix.

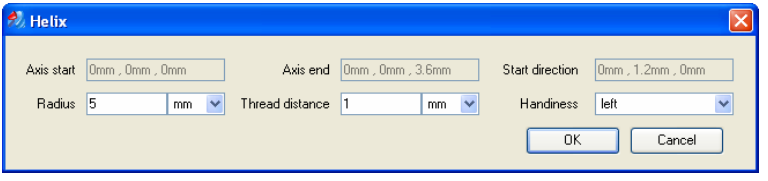
(2) Select the drawing plane X-Y, X-Z or Y-Z.

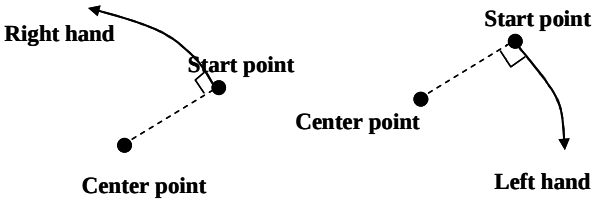
(3) Move the mouse icon to the initial point and the press the left mouse button to confirm the center point of bottom.

(4) Move the mouse icon along the axis of helix to draw a line and then press the left mouse button to confirm the axis of helix.

(5) Move the mouse icon to draw a line in the horizontal direction and then press the left mouse button to draw the radius of helix.

(6) Input the radius and thread distance in the **Helix** window.





Radius: radius of the helix.

Tread distance: distance between two adjacent layers.

Handiness: handiness of rotation direction of the helix.

(7) Click on the **OK** button to complete the drawing procedure.

(8) The geometry parameters are listed in the **Property** box and can be adjusted if necessary.

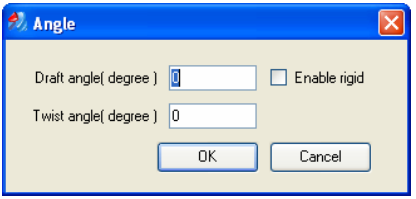
Next, we show how to draw a helix by combining the spline and helix options:

(1) Draw a helix;

(2) Draw a rectangle at the start point of helix, which is normal to the helix;

(3) Click on the  button, press and hold the **Ctrl** key on the keyboard, and then use the mouse icon to select the rectangle and helix.

(4) Click the button  in the **Toolbar**, and input the angle parameters in the following box:

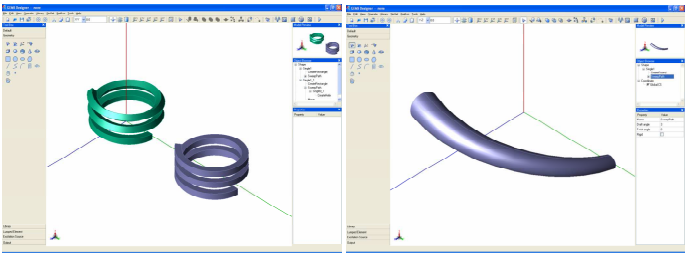


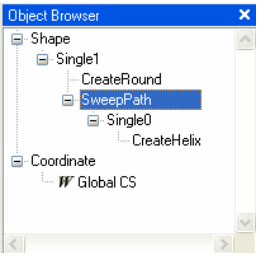
Draft angle: Control the variation of helix cross section.

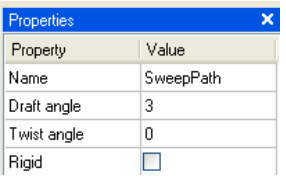
Twist angle: Control the rotation variation of helix cross section.

Enable rigid: End surface of helix is parallel to the system plane.

(5) Click on the **OK** button to draw a helix. Two types of helixes as shown below:







5.1.14 Spiral

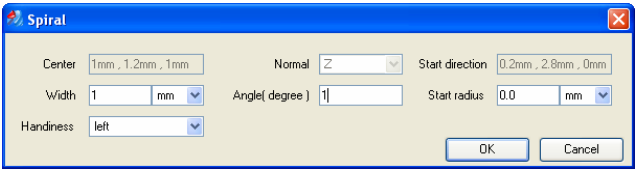
Steps to drawing a spiral are shown as follows:

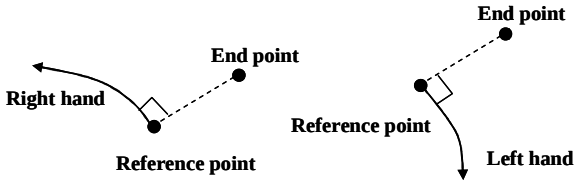
(1) Click on the icon in the **Tool Box**, move the mouse icon into the picture region, and select the center point of spiral.

(2) Select the drawing plane X-Y, X-Z or Y-Z.

(3) Move the mouse icon to the initial point and the press the left mouse button to confirm the center point.

(4) Move the mouse icon in the spiral plane to draw a line, and a dialogue window appears:





Width: Distance between adjacent spiral lines.

Normal: Normal direction of spiral.

Angle: Total rotation angle of spiral.

Start radius: Initial angle of spiral.

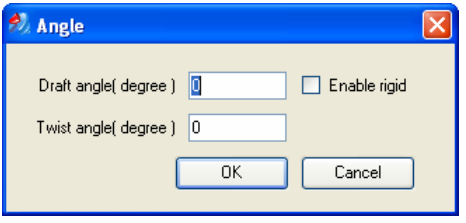
We use the spiral and rectangle to draw a spiral with finite size of cross section. The steps are listed as follows:

(1) Draw a spiral;

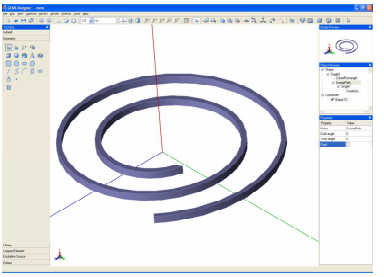
(2) Draw a rectangle at the start point of the spiral, which is perpendicular to the spiral;

(3) Click on the button, press and hold the **Ctrl** key on the keyboard, and then use the mouse icon to select the rectangle and spiral.

(4) Click the button in the **Toolbar**, and input the angle parameters in the following box:



Click the **OK** button to draw the spiral.



5.1.15 Spring

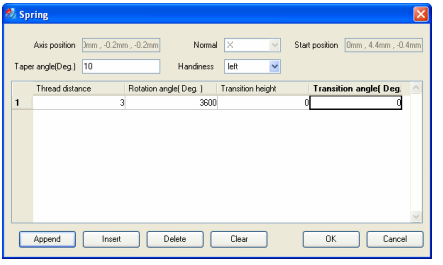
Steps to drawing a spring are shown as follows:

(1) Click on the icon  in the **Tool Box**, move the mouse icon into the picture region, and select the center point of spiral.

(2) Select the drawing plane  ,  or  .

(3) Move the mouse icon to the initial point and the press the left mouse button to confirm the center point.

(4) Move the mouse icon in the spring plane to draw a line, and a dialogue window appears:



Axis position: center point of the reference plane.

Normal: Axis direction of spring.

Start position: Start point of spring.

Taper angle: Taper angle of spring

Handiness: Rotation direction (left or right).

Thread distance: Distance between adjacent loops.

Rotation angle: Total rotation angle of spring.

Transition height: Transition height of two parts of springs

Transition angle: Angle of transition region.

Append: add a part of spring.

Inset: inset a part of spring.

Delete: delete a part of spring.

Clear: clear spring list.

Click the **OK** button to finish the spring drawing.

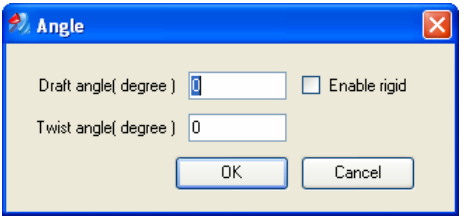
We use the spring and circle to draw a spring with finite size of cross section. The steps are listed as follows:

(1) Draw a spring;

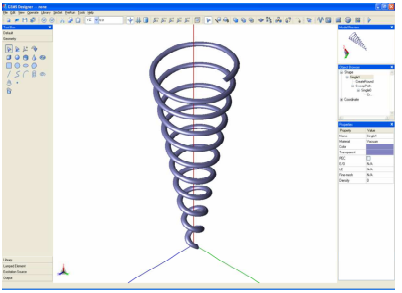
(2) Draw a circle at the start point of the spring, which is perpendicular to the spring;

(3) Click on the  button, press and hold the **Ctrl** key on the keyboard, and then use the mouse icon to select the circle and spring.

(4) Click the button  in the **Toolbar**, and input the angle parameters in the following box:



Click the **OK** button to draw a spring.



5.1.16 Point

Steps to drawing a dot are shown as follows:

(1) Click on the icon in the **Tool Box**, move the mouse icon into the picture region, and select the center point of spiral.

(2) Select the drawing plane X-Y, X-Z or Y-Z.

(3) Move the mouse icon to the desired point and the press the left mouse button.

5.2 Object Operations

In this section, we guide you to use the operations between the objects. It is very important for you to understand these operations to generate a beautiful and efficient GEMS project. All the objects inside the picture region have a corresponding entrance in the **Object Browser** box. You should first select the object before carrying any operations on it.

5.2.1 Delete Object

Delete an object by selecting the object in the figure region, or selecting an object in the **Shape** tree of **Object Browser** box, and then clicking on the  button, or pressing the **Delete** key on the keyboard.

5.2.2 Object Redo and Undo

Undo or redo an operation on an object by clicking on the  or  button in the toolbar, which allows you to undo or redo the operation on the object at the previous step.

5.2.3 Object Copy and Paste

GEMS allows you to copy an object in the same project or from one project to another one.

(1) Copy and paste an object by clicking on the  and then  button. This operation can also be carried out by first pressing and holding **Ctrl** + **c**, and then pressing and holding **Ctrl** + **v**. The pasted object is still at the position of the original object, so you need to move it to its own position by modifying its geometry coordinates in the **Properties** box. When an object is copied and pasted, its properties will be copied and pasted accordingly.

(2) Copy and paste an object from one project to another. In order to copy an object from one project to another, you need to open both the projects using two independent designer windows. Select the object that will be copied (use the  button or press and hold **Ctrl** + **c**) and then move the mouse icon to another project to paste the object into it (use the  button or press and hold **Ctrl** + **v**).

5.2.4 Object Moving

The operation of moving object allows you to move it from one place to another one. Follow the steps below to move an object:

(1) Select the object in the **Object Browser** box;

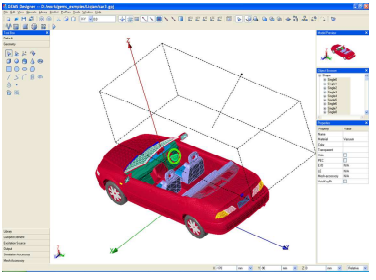
(2) Click on the button in the toolbar.

(3) Select a desired plane X-Y, X-Z or Y-Z, and a starting position;

(4) Move the mouse icon in the desired direction and distance, and click on the left mouse button. The selected object will be moved to the desired position.

(5) The moving distance is listed in the **Property** box and can be adjusted if necessary.

GEMS provides two ways to move an object or a group of objects, namely, (i) show the object details during the moving; (ii) show the minimum box only. The switch "Preview->fast preview" between (i) and (ii) are located in the **Set Default** option in the **Tools** menu.



5.2.5 Object Rotating

The operation rotation allows you to rotate an object from one orientation to another one. Follow the steps below to rotate an object:

(1) Select the object in the **Object Browser** box;

(2) Click on the button in the toolbar.

(3) Select a desired plane 

, and a starting position;

(4) Move the mouse icon in the desired direction and distance, and click on the left mouse button. The selected object will be rotated to the desired orientation.

(5) The rotation angle and axis are listed in the **Property** box and can be adjusted if necessary.

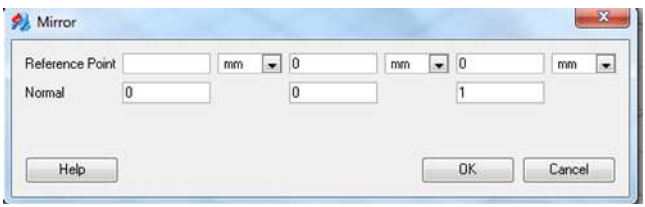
GEMS provides two ways to move an object or a group of objects, namely, (i) show the object details during the moving; (ii) show the minimum box only. The switch "Preview->fast preview" between (i) and (ii) are located in the **Set Default** option in the **Tools** menu.

5.2.6 Object Mirror

Follow the steps below to turn an object relative to the reference plane:

(1) Select an object;

(2) Click the button on the toolbar and following dialogue window appear:



Reference point: coordinates of the reference points.
Normal: normal direction of the reference plane passing the reference point.

The parameters in the dialogue are shown in the **Property** box and you can modify them if necessary.

5.2.7 Object Unite

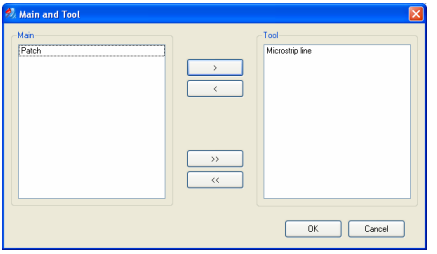
The unite operation allows you to unite the objects in to one. The united object has the same type of material. Follow the steps below to unite two objects:

(1) Select two objects in the **Object Browser** box;

(2) Click on the  button in the toolbar;

(3) Use the buttons >, >>, < and << to adjust the objects in the **Main** and **Tools** boxes.

(4) The united object has the same name as the object in the **Main** box.



.

Click on the **OK** button to finish the object unite. The unite history is listed in the **Shape** tree in the **Object Browser** box. You can split them to the original status by deleting the **Unite** option.

5.2.8 Object Subtraction

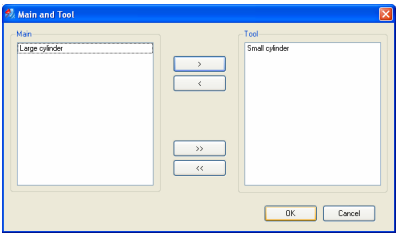
Follow the steps below to carry out the subtraction operation:

(1) Select at least two objects in the **Object Browser** box;

(2) Click on the  button in the toolbar;

(3) Use the buttons >, >>, < and << to adjust the objects in the **Main** and **Tools** boxes. The different part of object inside the **Main** box from the object inside the **Tools** box keeps inside the figure region.

(4) The left object has the same name as the object in the **Main** box.



Click on the **OK** button to finish the object subtraction. The subtraction history is listed in the **Shape** tree in the **Object Browser** box. You can recover objects by deleting the **Subtract** option.

5.2.9 Object Intersection

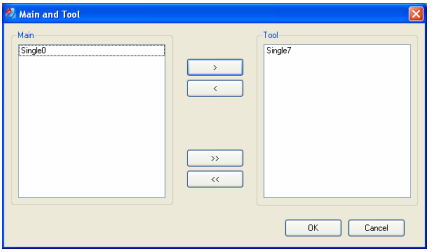
Follow the steps below to carry out the intersection operation:

(1) Select at least two objects in the **Object Browser** box;

(2) Click on the  button in the toolbar;

(3) Use the buttons >, >>, < and << to adjust the objects in the **Main** and **Tools** boxes. The overlapped part of objects inside the **Main** box with the object inside the **Tools** box keeps inside the figure region.

(4) The intersection part has the same name as the object in the **Main** box.



Click on the **OK** button to finish the object intersection. The intersection history is listed in the **Shape** tree in the **Object Browser** box. You can recover objects by deleting the **Intersection** option.

5.2.10 Object Group

Follow the steps below to group the selected object into one group:

(1) Select at least two objects in the **Object Browser** box;

(2) Click on the  button in the toolbar;

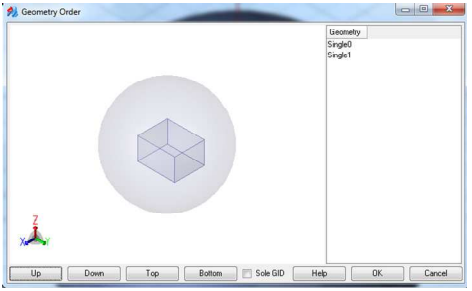
(3) The selected objects will be under one entrance inside the **Object Browser** box. You can split the group into the individual object by deleting the **Group** option.

5.2.11 Object Order Adjustment

In GEMS, the objects drawn later will erase the object drawn previously in the overlapping part. Follow the steps below to adjust the object order in the domain:

(1) Click on the  button in the toolbar;

(2) Adjust the object order by using the buttons at the bottom of the **Adjust order** window.



Up: move the selected object to upper position in the object list inside the **Object Browser** box.

Down: move the selected object to lower position in the object list inside the **Object Browser** box.

Top: move the selected object to top n the object list inside the **Object Browser** box.

Bottom: move the selected object to bottom in the object list inside the **Object Browser** box.

Sole GID: sort the index of excitation for the multiple ports. For example, there are 100 excitation ports, and each port is obtained by using the clone feature, to avoid changing the group ID 100 times, we can simply use this button to get an index order.

Click on the **OK** button to finish the object order adjust.

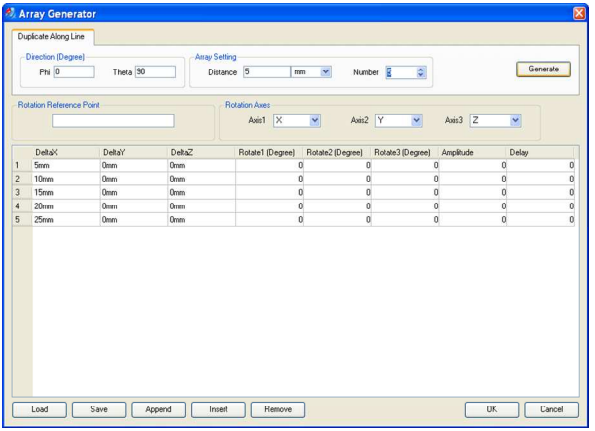
5.2.12 Object Array Generation

Follow the steps below to generate an array:

(1) Select one or more objects in the **Object Browser** box;

(2) Click on the  button in the toolbar;

(3) Input the array parameters in the following window:



Direction->Phi: Array orientation in the φ-direction

Direction->Theta: Array orientation in the θ-direction

Array Setting->Distance: Distance between the elements

Array Setting->Number: Number of elements

Generate: Generate an array

Rotation Reference point: Reference for the rotation

Rotation Axes->Axis 1: First rotation axis

Rotation Axes->Axis 2: Second rotation axis

Rotation Axes->Axis 3: Third rotation axis

DeltaX: Shift in the x-direction

DeltaY: Shift in the y-direction

DeltaZ: Shift in the z-direction

Rotate1(deg): Rotation angle around the first axis (θ1)

Rotate2(deg): Rotation angle around the second axis (θ2)

Rotate3(deg): Rotation angle around the third axis (θ3)

Amplitude: amplitude (for the excitation source)

Delay: time delay (for the excitation)

Load: Import an array data
Save: Store the array data in a text file
Clear: Clear the element list
Append: Add an element in the list
Insert: Inset an element in the list
Remove: Remove the selected element in the list

The element inside the array is the independent and can be operated as a regular object. If the array distribution is non-uniform, you can modify the array location inside the table. **GEMS** provides two ways to generate an array, namely, softclone and hardclone. The softclone operation is a clone technique in which the properties of the cloned objects vary with the original one. When you modify the original object or its attributes, all of the cloned objects change accordingly. However, the hardclone is a technique in which the cloned objects will be independent and they are not related to each other. You have to modify each of them whenever you want to.

You can generate an array through a data file following the format below:

(1) Array only includes the geometry objects without rotation.

```
0
x1  y1  z1
x2  y2  z2
x3  y3  z3
x4  y4  z4
x5  y5  z5
```

(2) Array only includes the geometry objects and excitation source without rotation.

```
10        (Array no rotation)
x1  y1  z1  amplitude1  time_delay1
```

x2 y2 z2 amplitude2 time_delay2
x3 y3 z3 amplitude3 time_delay3
x4 y4 z4 amplitude4 time_delay4
x5 y5 z5 amplitude5 time_delay5

(3) Array only includes the geometry objects and excitation source with rotation around one axis.

11
x0 y0 z0 axis (reference point and axis (0: x-axis; 1: y-axis; 2: z-axis)
x1 y1 z1 angle(deg.) amplitude1 time_delay1
x2 y2 z2 angle(deg.) amplitude2 time_delay2
x3 y3 z3 angle(deg.) amplitude3 time_delay3
x4 y4 z4 angle(deg.) amplitude4 time_delay4
x5 y5 z5 angle(deg.) amplitude5 time_delay5

(4) Array only includes the geometry objects and excitation source with rotation around two axes.

12
x0 y0 z0 axis1 axis2 (reference point and axis (0: x-axis; 1: y-axis; 2: z-axis)
x1 y1 z1 angle1(deg.) angle2(deg.) amplitude1 time_delay1
x2 y2 z2 angle1(deg.) angle2(deg.) amplitude2 time_delay2
x3 y3 z3 angle1(deg.) angle2(deg.) amplitude3 time_delay3
x4 y4 z4 angle1(deg.) angle2(deg.) amplitude4 time_delay4
x5 y5 z5 angle1(deg.) angle2(deg.) amplitude5 time_delay5

(5) Array only includes the geometry objects and excitation source with rotation around three axis.

13
x0 y0 z0 axis1 axis2 axis3 (reference point and axes)
x1 y1 z1 angle1(deg.) angle2(deg.) angle3(deg.) amplitude1 time_delay1
x1 y1 z2 angle2(deg.) angle2(deg.) angle3(deg.) amplitude1 time_delay2
x1 y1 z3 angle3(deg.) angle2(deg.) angle3(deg.) amplitude1 time_delay3

x1 y1 z4 angle4(deg.) angle2(deg.) angle3(deg.) amplitude1 time_delay4
x1 y1 z5 angle5(deg.) angle2(deg.) angle3(deg.) amplitude1 time_delay5

5.2.13 Object Sweeping

The line options such as the line, the spline, the spiral, the helix and the spring are usually used to generate a sweeping path for the sweeping options. The basic steps to use the sweeping option to draw a 3-D geometry are described below:

(1) Draw a 2-D or a 3-D object following the regular way.

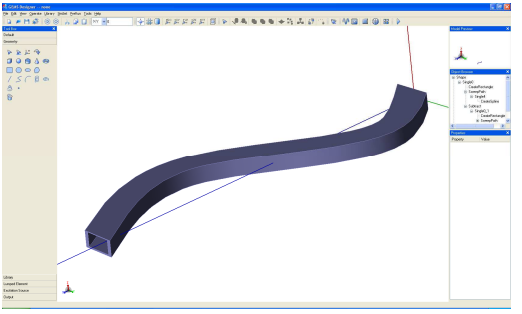
(2) Carry out any necessary object operations such as unite, subtract, intersect, or group.

(3) Draw a line, a spline, a spiral, a helix or a spring in the picture region starting from a position on \the object, which will be swept.

(4) Click on the  button and then select both the sweeping line and the object that will be swept by pressing and holding the **Ctrl** key on the keyboard.

(5) Click on the  button to realize the swept object.

If **GEMS** does not generate the swept object, please check the dimensions and make sure that the geometry does not have an ill-posed shape. A swept rectangular waveguide is shown in the window below.



5.2.14 Object Shelling

Shelling option is used to generate a shell with a finite thickness based on 3-D object. In addition, it can be used efficiently to generate some special geometries. Follow the steps below to carry out the shelling operation:

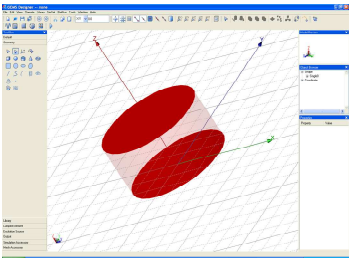
(1) Draw a cylinder;

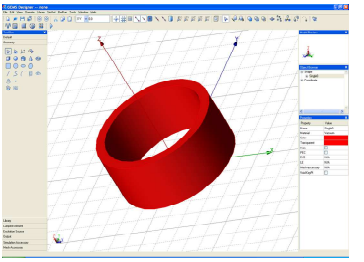
(2) Click the icon  inside the **Tool Box**;

(3) Select the top and bottom surfaces;

(4) Input the thickness of the guide in the following window:

Click on the **OK** button to finish the guide generation.





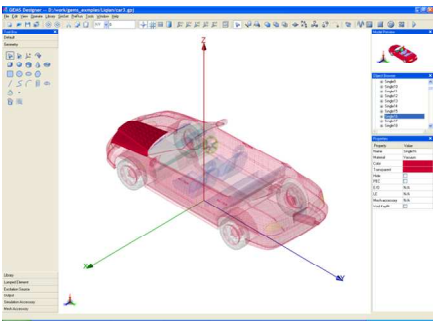
Follow the steps below to generate a dielectric layer on the car surface:

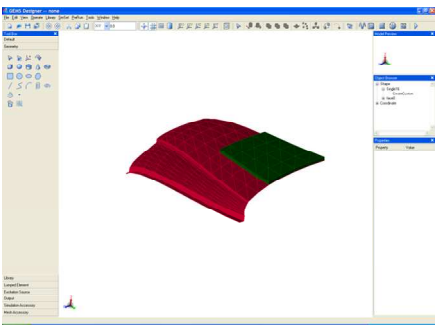
(1) Import a car model;

(2) Click the button  in the toolbar, and then select a surface on the car surface;

(3) Press **Ctrl+ c** and then **Ctrl + v** to replicate this surface;

(4) Select **Shelling** option in the **Operate** menu, and input the thickness of the dielectric layer.





5.2.15 Formula Transform

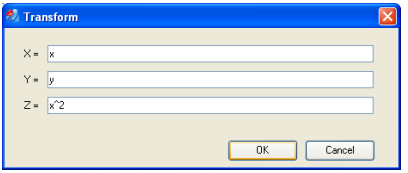
The formula transform option is designed to help you draw the special shapes of objects through an analytic way. Follow the steps below to draw a cylindrical surface through an analytic forma:

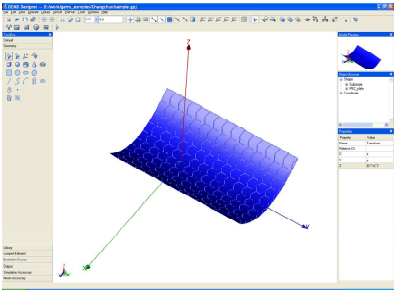
(1) Draw a 2-D surface or 3-D object structure;

(2) Click the icon  in the Toolbar ;

(3) Select the top and bottom surfaces.

Input the analytic formula of a cylinder in the following window. A cylindrical surface is generated.





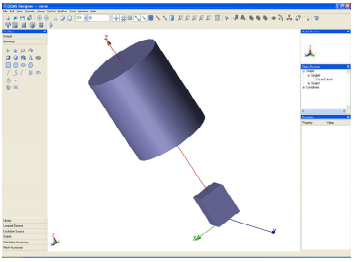
5.2.16 Loft Operation

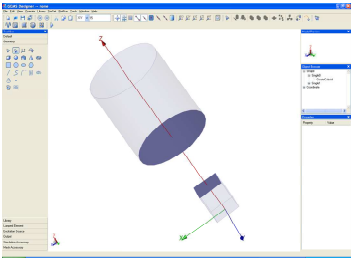
The Loft option is designed to help you draw the special shapes of objects. Following the steps given below to generate a horn antenna:

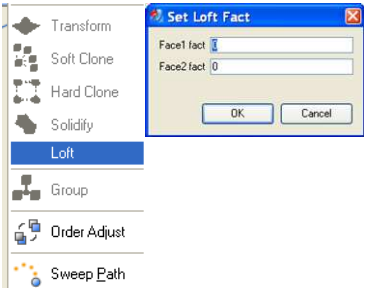
(1) Draw a cylinder and cuboid;

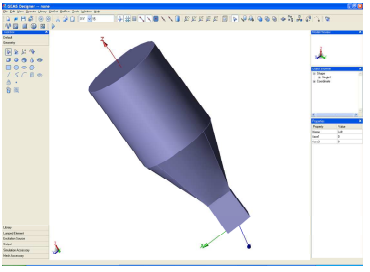
(2) Select the **Loft** option in the operate menu;

(3) Select two inner surfaces and input two parameters in the appeared window;









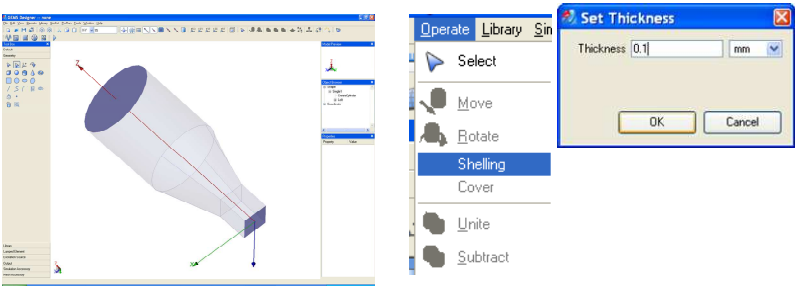
The parameters Facet1 factor and Facet2 factor controls the transition variation.

Combine with the shelling operation to generate a special waveguide. Following the steps given below to generate a waveguide:

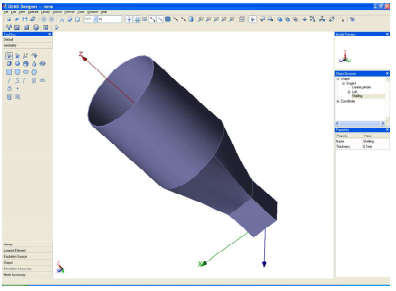
(1) Click on the surface selection icon in the tool box;

(2) select two outer surfaces;

(3) Select the **Shelling** option in the **Operate** menu, and specify the thickness of the waveguide in the appeared window.

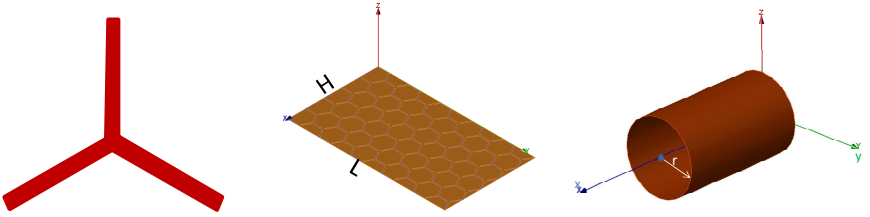


Click on the OK button to generate the waveguide.



5.2.17 Geometry Mapping

Map a planar structure to a cylindrical surface. The width and length of the planar structure are H and L, respectively. The axis of cylindrical structure that we will construct is along the *x*-direction. The radius of cylinder is *r*.



Follow the steps below to map the planar structure on a cylindrical surface:

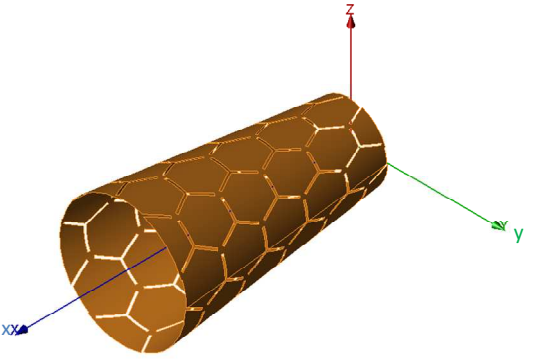
(1) Select the planar structure;

(2) Click the button  in the toolbar;

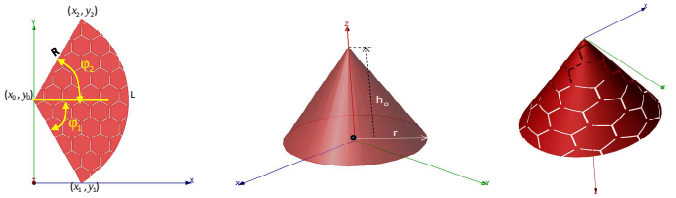
(3) Input the following formula in the appeared window:

$$z = \frac{L}{2\pi}\cos\left(2\pi\frac{x_0}{L}\right)$$
$$y = \frac{L}{2\pi}\sin\left(2\pi\frac{x_0}{L}\right)$$
$$x = y_0$$

(4) Select the *H* and *L* to be 50 mm and 86.6 mm， respectively.



We can get the following mapping using the similar mapping method:



The mapping relationship are:

$$x = r\cos(\varphi)\frac{h_0 - h}{h_0}$$
$$y = r\sin(\varphi)\frac{h_0 - h}{h_0}$$
$$z = h$$

where, the parameters are listed in the following table:

x_0	0 mm
y_0	43.3 mm
x_1	25 mm
y_1	0 mm
x_2	25 mm
y_2	86.6 mm
R	50 mm
L	116.35 mm
R	18.459 mm
H	46.468 mm
φ_2	1.047
φ_1	–1.047
φ	0 ~ 2π

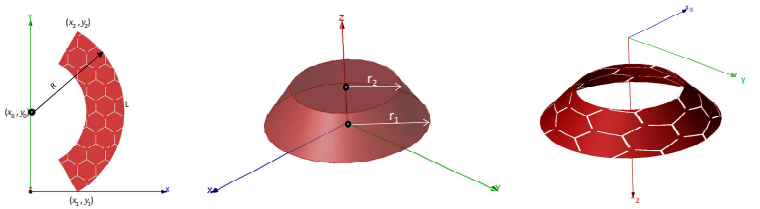
Using the following formulation:

x = cos((arctan((y-43.3)/x)+1.0455)/1.0455*π)*16.6389*sqrt(x*x+(y-43.3)*(y-43.3))/50

y = sin((arctan((y-43.3)/x)+1.0455)/1.0455*π)*16.6389*sqrt(x*x+(y-43.3)*(y-43.3))/50

z =(1-sqrt(x*x+(y-43.3)*(y-43.3))/50)

We can get the following mapping using the similar mapping method:



The parameters are listed in the following table:

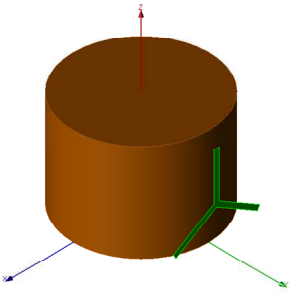
x_0	0 mm
y_0	43.3 mm
x_1	25 mm
y_1	0 mm
x_2	25 mm
y_2	86.6 mm
R	50 mm
L	116.35 mm
R_1	16.6389 mm
R_2	11.075 mm
H	46.468 mm
φ_2	1.055
φ_1	–1.055
φ	0 ~ 2π

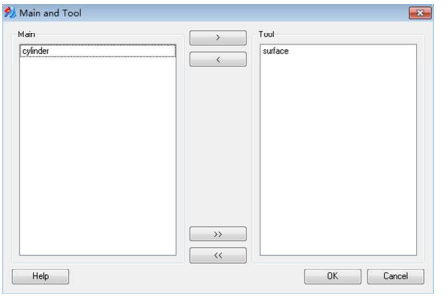
5.2.18 Project Operation

Project operation projects a planar or curved infinitely structure to the surface of a 3-D solid object. The two basic requirements: (1) the thin structure has no thickness; (2) the 3-D object must be solid.

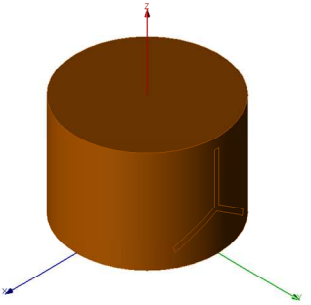
Follow the steps below to project a thin structure on to the surface of 3-D solid structure:

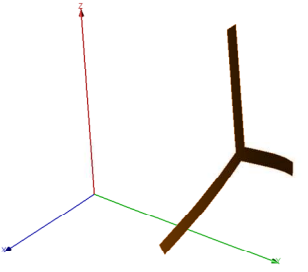
(1) Draw a 3-D structure;
(2) Draw a thin surface;
(3) Select the 3-D structure and the thin surface;
(4) Select the "Operate->Project" option;
(5) The 3-D solid structure should be in the **Main** box in the popup window, and the thin surface should be in the **Tool** box;
(6) Click the OK button to project the thin surface on to the surface of the 3-D solid object.





Select the "select face" icon in the "Tool Box->Geometry" box, and select the faces except the triple. And then delete the selected surfaces.





Chapter 6 Object Import and Export

The file formats that GEMS supports include ***.GOS** (GEMS format), ***.SAT** and **SAB** (**ACIS** format), ***.PYG** (polygon file), *.PYL (polyline file), ***.DXF** (**AutoCad** format), *.**STP**, *.**IGS**, and **Voxel**.

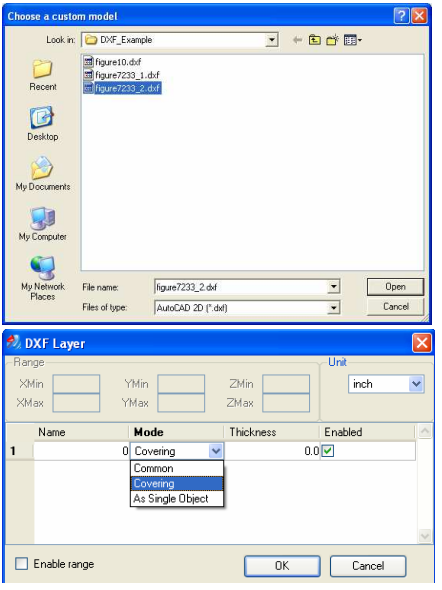
6.1 DXF File

Follow the steps below to import a DXF file:

(1) Click on the  button in the **Geometry** box;

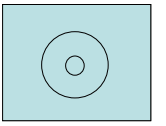
(2) Select the file format **AutoCad** 2D (***.dxf**) from the file type list;

(3) Search the **AutoCad** file (AutoCAD 2004 format only). You will see the following window if the file format is correct.

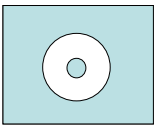


There are 3 options in the **Mode** box, namely, **Common**, **Covering**, and **As Single Object**. The **Common** option indicates that GEMS will load the original design regardless of the drawing types in the **AutoCad**. The **Covering** option will fill-in the polygonal lines as a solid polygon (plate). The **As Single Object** option will treat the separated objects as a single one. A simple example (a via in a PCB design) is graphically shown below:

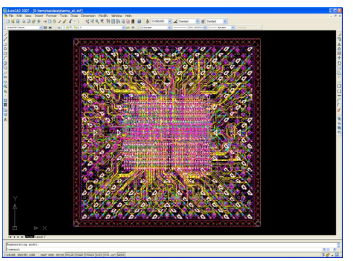
3 objects in the original design:

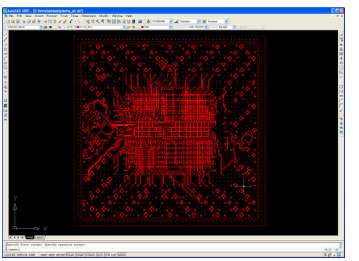


A single object imported into GEMS:



Next, we use a practical example to demonstrate how to import a complex **AutoCad DXF** file into the GEMS interface.



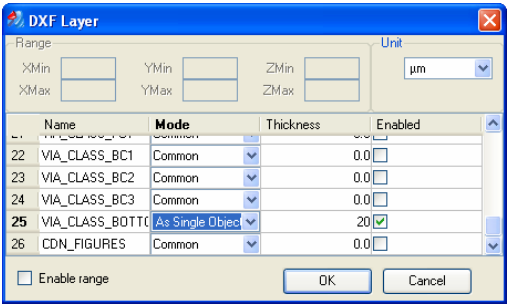


(1) Click on the button in the **Geometry** box;

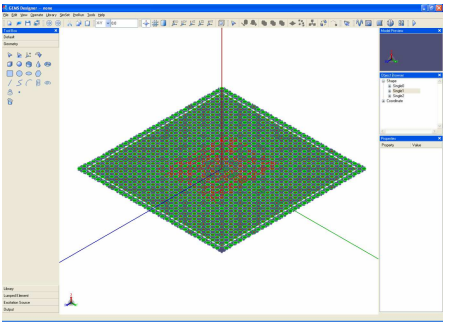
(2) Select the file format **AutoCad** 2D (***.dxf**) from the file type list;

(3) Search the **AutoCAD** file. You will see the following window if the file format is correct.

All the layers of the chip will be listed in the window below.



GEMS provides the following options for you: (i) selection of fill-in format of the **AutoCAD** drawing (**Common**, **Covering** and **As Single Object**); (ii) layer thickness including the ground, via, pins and circuits; (iii) number of the layers; (iv) the range of interest. For example, if we select to import the first ground layer, the first pin layer and via layer of the complete chip, GEMS generates the structure shown in the window below:



The middle layer and the circuit layer are shown in the GEMS window below:

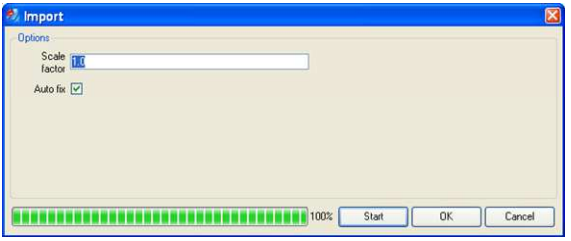
6.2 SAT File

Follow the steps below to import a SAT file to GEMS interface:

(1) Click on the  button in the **Geometry** box;

(2) Select the file format **SAT** from the file type list;

(3) Search the SAT file (*.sat). You will see the following window if the file format is correct.



Scale factor: scale factor that can enlarges or reduces the model size.

Auto fix: Check it to fix the minor problems in the model.

Start: Start the import process.

OK: Close the window.

(4) Click on the **OK** button to finish the loading procedure.

6.3 STEP File

Follow the same procedure as the SAT format, you can import a STEP model into the GEMS interface.

6.4 ProE Data

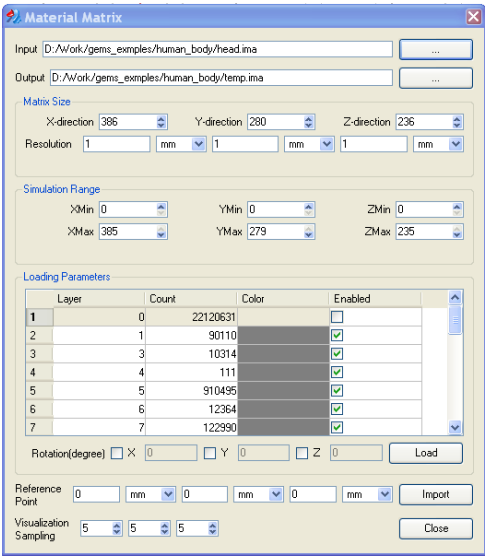
Follow the same procedure as the SAT format, you can import the ProE model into the GEMS interface.

6.5 Voxel Data

Follow the steps below to import Voxel model to GEMS interface:

(1) Select Material Matrix option in the Tools menu;

(2) Select Voxel (*.ima) file, input the parameters in the following window:



Input: load the original Voxel file（extension name is ima）

Output: Output file path and name

Matrix size->X-direction: Size of the original model in x-direction

Matrix size->Y-direction: Size of the original model in y-direction

Matrix size->Z-direction: Size of the original model in z-direction

Matrix size->Resolution: Resolution of matrix in the original model

Simulation Range->XMin: Initial value of the simulation range in the x-direction

Simulation Range->XMax: End value of the simulation range in the x-direction

Simulation Range->YMin: Initial value of the simulation range in the y-direction

Simulation Range->YMax: End value of the simulation range in the y-direction

Simulation Range->ZMin: Initial value of the simulation range in the z-direction

Simulation Range->ZMax: End value of the simulation range in the z-direction

Loading Parameters->Layer: Number of layers in the model
Loading Parameters->Count: Number of Voxel elements in the model
Loading Parameters->Color: Color of dielectric layer
Loading Parameters->Enabled: Check it to enable the layer
Loading Parameters->Rotation->X: Rotation angle about the x-axis
Loading Parameters->Rotation->Y: Rotation angle about the y-axis
Loading Parameters->Rotation->Z: Rotation angle about the z-axis
Loading Parameters->Load: Load the data into table.

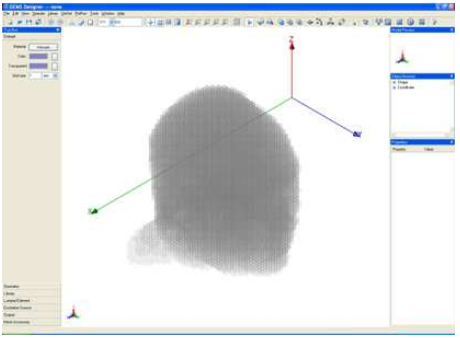
Reference Point->(X): Reference in the x-direction
Reference Point->(Y): Reference in the y-direction
Reference Point->(Z): Reference in the z-direction

Import: Click to import the model to GEMS

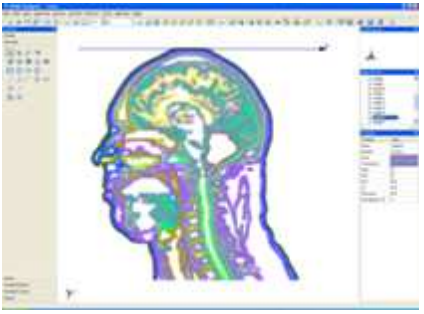
Visualization Sampling->(X): Display resolution in the x-direction
Visualization Sampling->(Y): Display resolution in the y-direction
Visualization Sampling->(Z): Display resolution in the z-direction
（Reduce the display number but does not affect the simulation result.）
Close: Close window

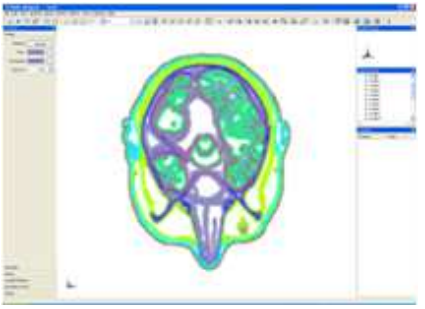
(3) Click the **Load** button and then select display color, reference point and resolution etc..

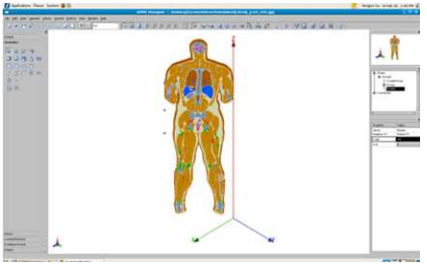
(4) Click the **Import** button to import the model to GEMS.

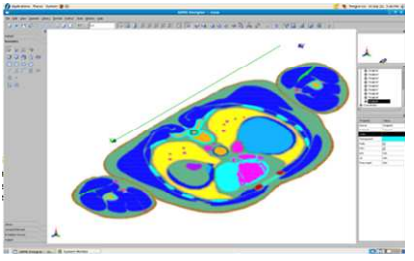


You can view the model slice by slice and change the color of each layer of tissue. The several cuts of human head and entire human body are displayed in the figures below.









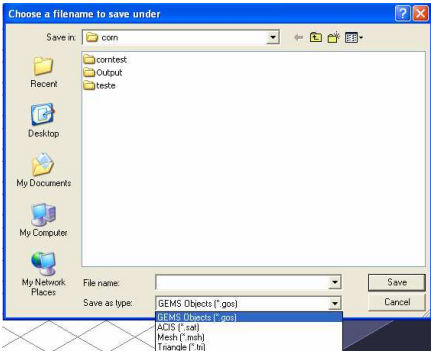
6.6 GEMS File Export

GEMS can export the Standard **ACIS** Text (SAT) and Standard **ACIS** Binary (SAB) file formats. Follow the steps below to export the model:

(1) Select the structure you like to export in the picture region;

(2) Select the **Export** option in the **File** menu;

(3) Select the ACIS (*sat) option in the following window, and then specify a file name.



In addition, GEMS can export the structure in other several different file formats, namely, *.gos (GEMS format), *.msh (GID format), and *.tri (CFDTD format). GEMS also provides an alternative option, **Export Separately**, which allows you to export the selected structures into a file, and each of them is independent when they are imported back to the GEMS interface.

Chapter 7 GEMS Project Design

In this chapter, we describe the major procedure to create a GEMS project. A valid GEMS project should include at least one excitation source and output, the boundary condition, and the valid mesh distribution.

7.1 Types of Excitation Pulse

The excitation pulse design is by clicking the button  in the toolbar. GEMS provides the following type of excitation pulses:

(1) pure **Gaussian** (input parameter: frequency of 3dB bandwidth)

(2) **Differential Gaussian** (input parameter: frequency of 3dB bandwidth)

(3) **Modulated Gaussian** (input parameters: frequency of 3dB bandwidth and modulated frequency)

(4) Sinusoid wave (input parameters: frequency)

(5) Square wave (input parameters: period, number of pulse cycles, raise time, hold time, and fall time);

(6) Self-defined pulse and the file has the following format:

Δt (time sampling space in the user defined pulse, the unit is nanosecond)
Value at t=0 (initial value of the pulse)
Value at t=Δt
Value at t=2Δt
Value at t=3Δt
Value at t=4Δt
:

:

:

Value at t=(n-1)Δt

Value at t=nΔt

7.2 Excitation Type

GEMS provides a large number of excitation sources for the applications to antennas, microwave circuits, antenna array, EBG structures, frequency selective surfaces (FSS), plane wave source. We begin with the lumped excitation source.

7.2.1 Lumped port excitation

The lumped port source  combines a voltage source, lumped element, voltage output and current output. The lumped port is usually used to excite an open port. For an antenna problem, this port will return the return loss, input impedance, time voltage and current. For a circuit problem, this port will return the s_{11}, port impedance, time port voltage and current. A lumped port excitation has the following appearance.

Follow the steps given below to add a lumped port excitation:

(1) Click on the line segment  or line  in the **Tool Box**.

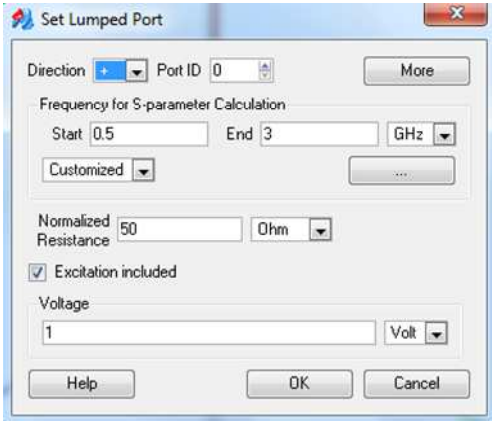
(2) Select the drawing plane (X-Y, Y-Z or X-Z).

(3) Specify the position in the third axis.

(4) Draw a line from the ground to the signal conductor.

(5) Select the option in the Object Browser box.

(6) Click on the  button in the tool box.



Direction: Black end touches the positive pole

Port ID: It is no meaning for the single port. For the multiple ports, the ports with the same ID number, will be excited at the same time.

More: For more port options.

Frequency for S-parameter Calculation->Start: Initial frequency

Frequency for S-parameter Calculation->End: End frequency

Frequency for S-parameter Calculation->(unit): Frequency unit

Frequency for S-parameter Calculation->Customized): Self-defined number of frequency sampling.

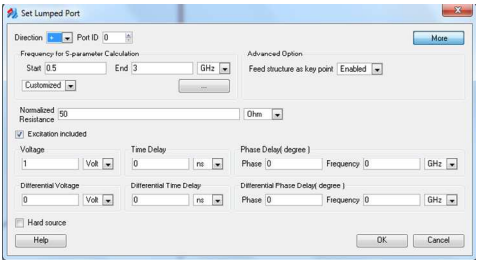
Frequency for S-parameter Calculation->(Adaptive): Default number of frequency sampling.

Normalized Resistance: Internal resistance of the excitation source, S-parameter is normalized to this value.

Excitation included: Uncheck it for the output port.

Voltage: Amplitude of excitation source.

Help: Open help file; **OK**: Confirm the settings; **Cancel**: Cancel the settings.



Time Delay: The excitation pulse is launched after the certain time period, which is specified in the **Time Delay** box.

Phase delay->phase: Time delay at the specified frequency

Phase delay->frequency: Frequency of interest

Differential voltage: Differential voltage

Differential time delay: Differential time delay

Differential Phase Delay->Phase: Time delay of the differential voltage at the specified frequency

Differential Phase Delay->Frequency: Frequency of interest.

Advanced Option->Feed structure as key point: Allow the code to find the key points of the feed structure, which touches the positive pole of the lumped port when this option is enabled.

The value in the **Differential Voltage** shows the combination of two excitation pulses, for example, the first and second excitation pulses are expressed as $h_1(t)$ and $h_2(t)$, respectively. The differential pulse is defined as $h_1(t) - h_2(t)$. If the value in the **Differential Voltage** box is "0", $h_2(t) = 0$. Otherwise, $h_2(t) = \alpha\, h_1(t\text{-}t_0)$ at the

selected frequency. The Differential delay box will become active when the value is not zero in the **Differential Voltage** box.

7.2.2 Wave Port Excitation

The wave port source combines a voltage source, voltage output and current output. The wave port is usually used to excite a matched port. For a circuit problem, this port will return the s_{11}, port impedance, time port voltage and current. A wave port excitation has the following appearance.

Follow the steps given below to add a wave port excitation:

(1) Click on the line segment or the line in the **Tool Box**.

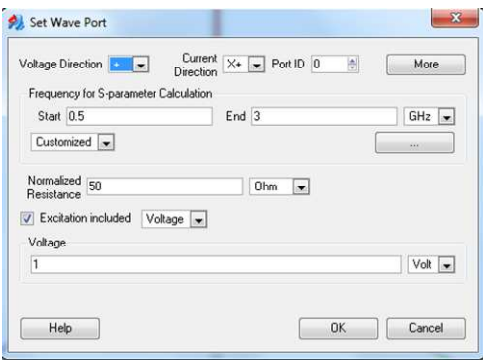
(2) Select the drawing plane (X-Y, Y-Z or X-Z).

(3) Specify the position in the third axis.

(4) Draw a line from the ground to signal conductor.

(5) Select the option in the **Object Browser** box.

(6) Click on the button in the **Tool Box**.



Direction: Black end touches the signal line, and the black line points to the wave propagation direction.

Port ID: It is no meaning for the single port. For the multiple ports, the ports with the same ID number, will be excited at the same time.

More: For more port options.

Frequency for S-parameter Calculation->Start: Initial frequency

Frequency for S-parameter Calculation->End: End frequency

Frequency for S-parameter Calculation->(unit): Frequency unit

Frequency for S-parameter Calculation->Customized): Self-defined number of frequency sampling.

Frequency for S-parameter Calculation->(Adaptive): Default number of frequency sampling.

Normalized Resistance: Internal resistance of the excitation source, S-parameter is normalized to this value.

Excitation included: Uncheck it for the output port.

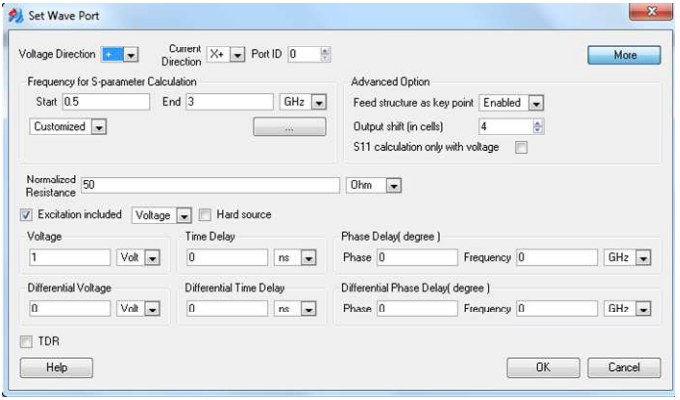
Voltage: Amplitude of excitation source.

Differential mode excitation: set two wave ports have same ID and opposite voltage amplitude.

Common mode excitation: set two wave ports have same ID and same voltage amplitude.

Help: Open help file; **OK**: Confirm the settings; **Cancel**: Cancel the settings.

The value in the **Differential Voltage** shows the combination of two excitation pulses, for example, the first and second excitation pulses are expressed as $h_1(t)$ and $h_2(t)$, respectively. The differential pulse is defined as $h_1(t) - h_2(t)$. If the value in the **Differential Voltage** box is "0", $h_2(t) = 0$. Otherwise, $h_2(t) = \alpha\, h_1(t\text{-}t_0)$ at the selected frequency. The Differential delay box will become active when the value is not zero in the **Differential Voltage** box.



Time Delay: The excitation pulse is launched after the certain time period, which is specified in the **Time Delay** box.

Phase delay->phase: Time delay at the specified frequency

Phase delay->frequency: Frequency of interest

Differential voltage: Differential voltage

Differential time delay: Differential time delay

Differential Phase Delay->Phase: Time delay of the differential voltage at the specified frequency

Differential Phase Delay->Frequency: Frequency of interest.

Advanced Option->Feed structure as key point: Allow the code to find the key points of the feed structure, which touches the positive pole of the lumped port when this option is enabled.

7.2.3 Voltage Excitation

Follow the steps given below to draw a voltage excitation source:

(1) Click on the line object in the **Tool Box**.

(2) Select the drawing plane (X-Y, Y-Z or X-Z).

(3) Specify the position in the third axis.

(4) Draw a line from the ground to signal line.

(5) Select the option in the Object Browser box.

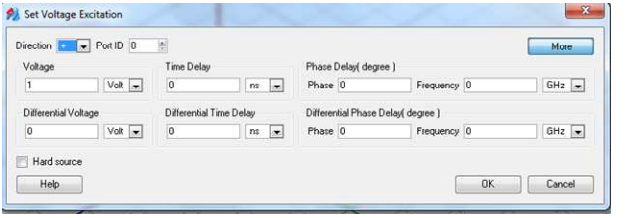
(6) Click on the  button in the tool box.

Direction: Black end touches the positive pole.

Port ID: It is no meaning for the single port. For the multiple ports, the ports with the same ID number, will be excited at the same time.

More: For more port options.

Click on the **More** button to open a new window.



Time Delay: The excitation pulse is launched after the certain time period, which is specified in the **Time Delay** box.

Phase delay->phase: Time delay at the specified frequency

Phase delay->frequency: Frequency of interest

Differential voltage: Differential voltage

Differential time delay: Differential time delay

Differential Phase Delay->Phase: Time delay of the differential voltage at the specified frequency

Differential Phase Delay->Frequency: Frequency of interest.

7.2.4 Current Excitation

Follow the steps given below to draw a voltage excitation source:

(1) Click on a 2-D surface icon such as rectangle, circle, or polygon in the **Tool Box**.

(2) Select the drawing plane (X-Y, Y-Z or X-Z).

(3) Specify the position in the third axis.

(4) Draw a surface that has the same size as the signal conductor, or larger.

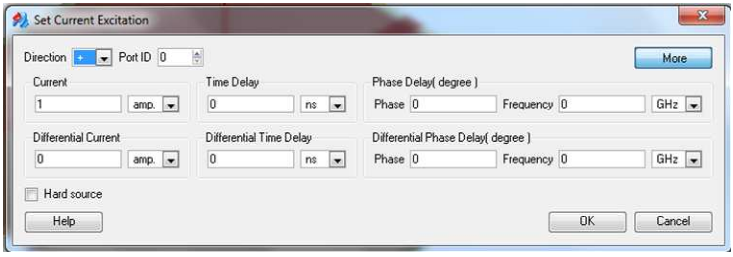
(5) Select the option in the **Object Browser** box.

(6) Click on the  button in the **Tool Box**.

Direction: Black end touches the positive pole.

Port ID: It is no meaning for the single port. For the multiple ports, the ports with the same ID number, will be excited at the same time.

More: For more port options.



Time Delay: The excitation pulse is launched after the certain time period, which is specified in the **Time Delay** box.

Phase delay->phase: Time delay at the specified frequency

Phase delay->frequency: Frequency of interest

Differential current: Differential current

Differential time delay: Differential time delay

Differential Phase Delay->Phase: Time delay of the differential current at the specified frequency

Differential Phase Delay->Frequency: Frequency of interest.

7.2.5 Mode Excitation

Follow the steps below to add a mode excitation:

(1) Click on a 2-D surface icon such as rectangle, circle, or polygon in the **Tool Box**.

(2) Select the drawing plane (X-Y, Y-Z or X-Z).

(3) Specify the position in the third axis.

(4) Draw a 2-D surface that has the same size as the inner cross section of the guide.

(5) Select the option in the **Object Browser** box.

(6) Click on the  button in the **Tool Box**.

Direction: The line with a black end points to the wave propagation direction.

Port ID: It is no meaning for the single port. For the multiple ports, the ports with the same ID number, will be excited at the same time.

More: For more port options.

Frequency for S-parameter Calculation->Start: Initial frequency

Frequency for S-parameter Calculation->End: End frequency

Frequency for S-parameter Calculation->(unit): Frequency unit

Frequency for S-parameter Calculation->Customized): Self-defined number of frequency sampling.

Frequency for S-parameter Calculation->(Adaptive): Default number of frequency sampling.

Port boundary truncation->XMin: Boundary at the x minimum direction.

Port boundary truncation->XMax: Boundary at the x maximum direction.

Port boundary truncation->YMin: Boundary at the y minimum direction.

Port boundary truncation->YMax: Boundary at the y maximum direction.

Port boundary truncation->ZMin: Boundary at the z minimum direction.

Port boundary truncation->ZMax: Boundary at the z maximum direction.

For the symmetric structure, the proper boundary is selected in this window.

Amplitude: Amplitude of the mode excitation in the time domain.

Matched load included: Check it for the open mode port.

Frequency range->Minimum: Pre-set initial frequency

Frequency range->Maximum: Pre-set end frequency

One port structure may support many modes, you can be only interested in the modes inside the frequency band of interest.

Mode Pattern->Analytic: For the rectangular and circular waveguide port, check this option will speed up the mode extraction.

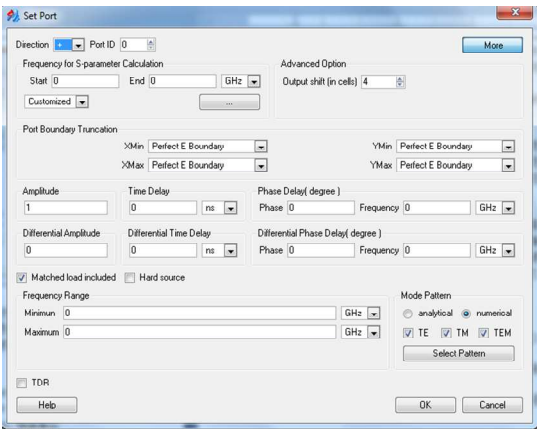
Mode Pattern->Numerical: All other port structures, you need to check the **Numerical** button to get the correct results.

Mode Pattern->TE: Mode of interest is TE

Mode Pattern->TM: Mode of interest is TM

Mode Pattern->TEM: Mode of interest is TEM

Mode Pattern->Select Pattern: open the mode extraction window. Click the More button to set more parameters.



Differential voltage: Amplitude of the differential voltage

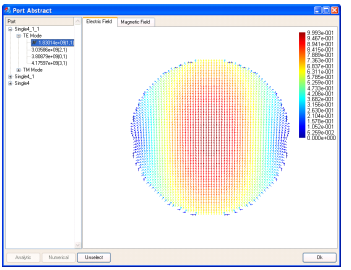
Differential time delay: Differential time delay

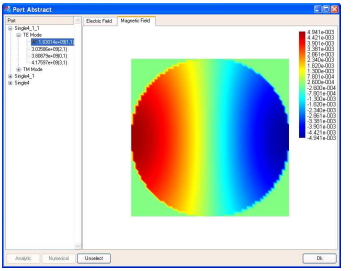
Differential Phase Delay->Phase: Phase delay for the differential voltage.

Differential Phase Delay->Frequency: Frequency of interest.

TDR： Check this option for TDR output. In this case, the excitation pulse should be a narrow pure Gaussian, and the simulation can be terminated after the pulse travels through the problem structure. If you are interested in the frequency domain results, the simulation should be convergent.

If the mesh distribution has been designed, click on the button  in the toolbar or, on the button **Set Pattern** to extract the port mode.





Follow the steps below to extract the port mode:

(1) Select one port in the port list.

(2) Click on the Analytic button to extract the mode pattern for the rectangular or circular cross section guide.

(3) The mode with its cut-off frequency will be listed in the mode tree.

(4) Select the mode that will be used as the excitation.

(5) Click on the **Select** button.

(6) Click on the title **Electric Field** or **Magnetic field** to view the mode pattern.

(7) Click on the **OK** button to return to the designer.

7.2.6 Aperture Field Excitation

Aperture field excitation is a spatial distribution on the specified region, which may be an ideal, measured or simulated field distribution, and uniform or non-uniform distribution. Follow the steps below to add an aperture field excitation:

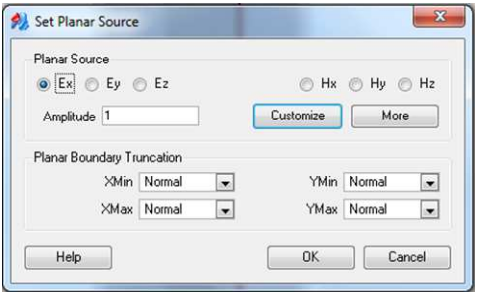
(1) Draw a rectangle or circle;

(2) Select the object inside the **Object Browser** box;

(3) Change its name in the **Property** box;

(4) Specify the boundary condition and mesh distribution etc., and then set the aperture field excitation;

(5) Click on the button in the Excitation Source box, and set the parameters inside following window:



Planar Source->Ex, Ey, Ez, Hx, Hy, Hz: Polarization

Planar Source->Amplitude: Amplitude of the excitation

Planar Source->Customize: Load a self-defined distribution

Planar Source->More: Set more excitation parameters

Port boundary truncation->XMin: Boundary at the x minimum direction.

Port boundary truncation->XMax: Boundary at the x maximum direction.

Port boundary truncation->YMin: Boundary at the y minimum direction.

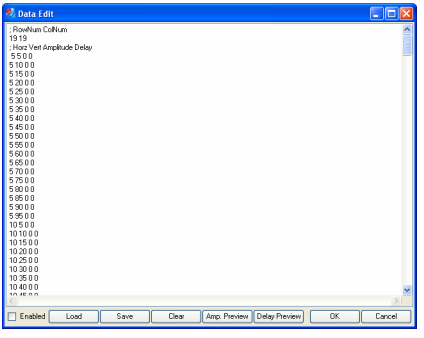
Port boundary truncation->YMax: Boundary at the y maximum direction.

Port boundary truncation->ZMin: Boundary at the z minimum direction.

Port boundary truncation->ZMax: Boundary at the z maximum direction.

The boundary of the aperture field distribution includes **Normal** and **Mirror**. For the symmetric structure, the proper boundary is selected in this window.

Click on the button **Customize** to open a new window to set the aperture field excitation. The first and second numbers in the first row represent the numbers of rows and columns, respectively. The first two numbers in each followed line includes four numbers, which are horizontal and vertical coordinates, respectively, namely, x: in the X-Y plane, y: in the Y-Z plane, z: in the Z-X plane; y: in the X-Y plane, z: in the Y-Z plane, x: in the Z-X plane. The followed are amplitude and phase, respectively.



Enabled: Check to activate the aperture field excitation

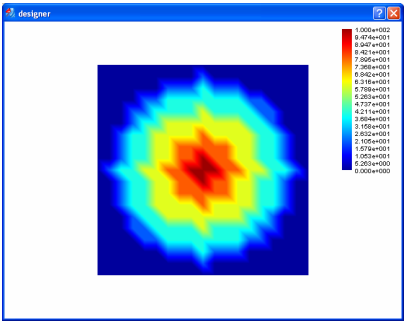
Load: Import the field distribution as the excitation

Save: Store the field distribution into a file

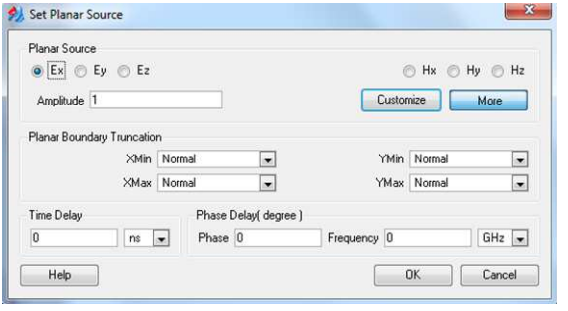
Clear: Clear the data inside the window

Amp Preview: Pre-view the field distribution

Phase Preview: Pre-view the phase distribution



Click on the **More** button to set more parameters in the following window:



Time Delay: Time delay of the excitation

Phase Delay->Phase: Phase delay at the specified frequency

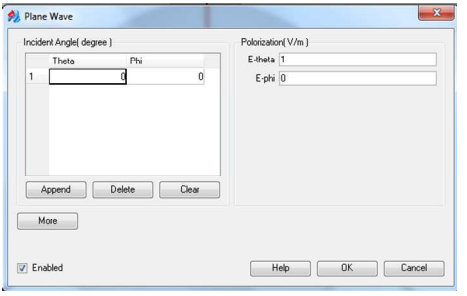
Phase Delay->Frequency: Frequency of interest

7.2.7 Plane Wave Excitation

Follow the steps below to add a plane wave excitation:

(1) Specify the excitation pulse, boundary condition and design the mesh distribution;

(2) Click on the button  in **Excitation Source** box of the **Tool Box**. Specify the plane wave parameters in the following window:



Enabled: Check it to activate this plane wave source.

Incident Angle->Theta: Incident angle θ

Incident Angle->Phi: Incident angle φ

Incident Angle->Append: Add a plane wave source

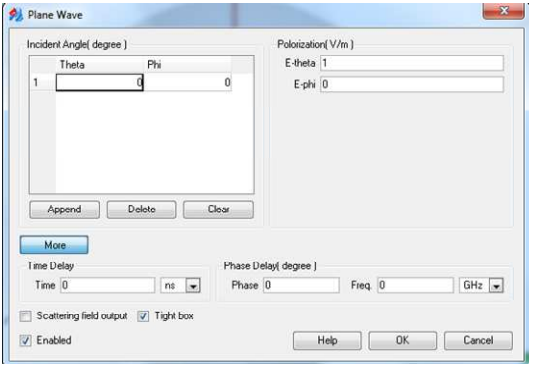
Incident Angle->Delete: Delete a plane wave source

Incident Angle->Clear: Clear the plane wave list

Polarization->E_Theta: θ-polarization component

Polarization->E_Phi: φ-polarization component

Click on the **More** button to set more plane wave parameters:



Scattering field output: Check it to output the scattered field (near field). The far field is always is scattered field.

Time Delay: Time delay of plane wave excitation.

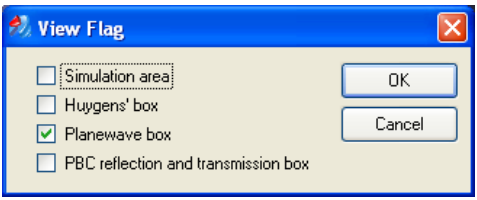
Phase Delay->Phase: Phase delay.

Phase Delay->Frequency: Frequency of interest.

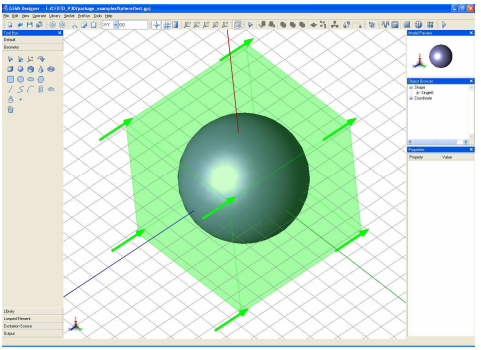
Tight Box: GEMS reserved.

The plane wave source is not related to a geometry. However, its relative location inside the domain can be viewed by clicking the button  in the toolbar.

.



Select the option **Plane wave box** to view the polarization and incident direction of the plane source.



7.3 Boundary Condition Setting

GEMS provides the following types of boundary conditions:

(1) E boundary. The tangential components of the eclectic field are zero on it.

(2) H boundary: The tangential components of the magnetic field are zero on it.

(3) Open boundary (touched): Absorbing boundary touches the geometry.

(4) Open boundary (add space): Absorbing boundary has a white space with the geometry.

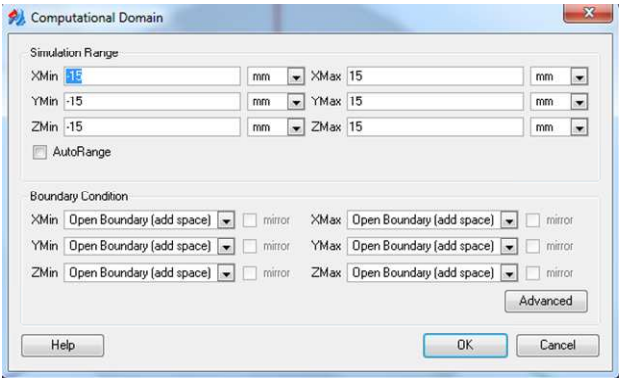
(5) Periodic boundary condition (PBC): Truncate the periodic structure.

(6) Mirror boundary condition: Simulate the symmetric structure using the E or H boundary and the far field parameters are output options.

Follow the steps below to specify the boundary condition:

(1) Specify the excitation pulse;

(2) Click on the button  in the toolbar, and then specify the boundary condition in the following window:



Simulation Range-XMin: The start coordinate of simulation domain in the x-direction.

Simulation Range-XMax: The end coordinate of simulation domain in the y-direction.

Simulation Range-YMin: The start coordinate of simulation domain in the y-direction.

Simulation Range-YMax:The end coordinate of simulation domain in the y-direction.

Simulation Range-ZMin: The start coordinate of simulation domain in the z-direction.

Simulation Range-ZMax:The end coordinate of simulation domain in the z-direction.

AutoRange: Check it to make the boundary touch the objects. You can specify them manually if it is necessary.

Boundary Condition->XMin: Boundary type in the -x-direction.

Boundary Condition->XMax: Boundary type in the +x-direction.

Boundary Condition->YMin: Boundary type in the -y-direction.

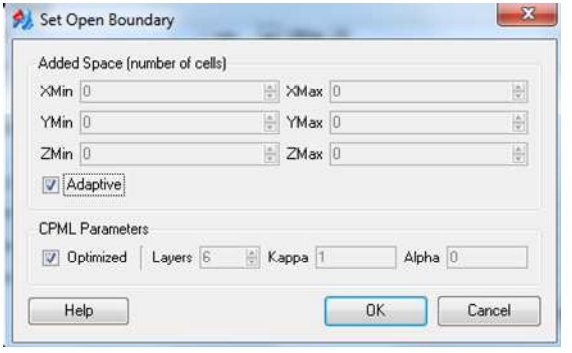
Boundary Condition->YMax: Boundary type in the +y-direction.

Boundary Condition->ZMin: Boundary type in the -z-direction.

Boundary Condition->ZMax: Boundary type in the +z-direction.

Mirror: If the E or H boundary condition is selected and the far field is output parameter, check it to generate the correct far fields. The periodic boundary condition can be only applied in the X-Y plane.

Click on the **Advanced** button to set more parameters:



Added Space->XMin: White space in cells in the –x-direction.

Added Space->XMax: White space in cells in the +x-direction.

Added Space->YMin: White space in cells in the –y-direction.

Added Space->YMax: White space in cells in the +y-direction.

Added Space->ZMin: White space in cells in the –z-direction.

Added Space->ZMax: White space in cells in the +z-direction.

Adaptive: GEMS adapts the white space based on the domain feature. You can manually change it if necessary.

CPML->Optimized: Check it to allow GEMS to select the PML parameters.

CPML->Layers: Default number is 6, and the maximum number is 10.

CPML->Kappa: Default number is 1, and it is taken to be a larger number if the simulation is unstable.

CPML->Alpha: Default number is 0, and the maximum number is 2 to improve the PML performance at the low frequencies.

7.4 GEMS Mesh Design

Mesh design requires completing the excitation pulse and boundary condition. Follow the steps below to design the mesh distribution:

(1) Specify the excitation pulse and boundary condition;

(2) Click on the button  in the toolbar, and design the mesh distribution in the following window:

Reference Frequency: Maximum frequency of interest.

Relative Permittivity: Maximum relative dielectric constant.

Relative Permeability: Maximum relative dielectric permeability.

X-Direction->MinSize: Minimum cell size in the x-direction (it is usually smaller than the fine structure of interest).

X-Direction->Ratio: Ratio of adjacent cell size. (usually is smaller than 2.5)

Y-Direction->MinSize: Minimum cell size in the y-direction (it is usually smaller than the fine structure of interest).

X-Direction->Ratio: Ratio of adjacent cell size. (usually is smaller than 2.5)

Z-Direction->MinSize: Minimum cell size in the z-direction (it is usually smaller than the fine structure of interest).

X-Direction->Ratio: Ratio of adjacent cell size. (usually is smaller than 2.5)

Checking the **Uniform** box will ignore all the geometry during the mesh generation.

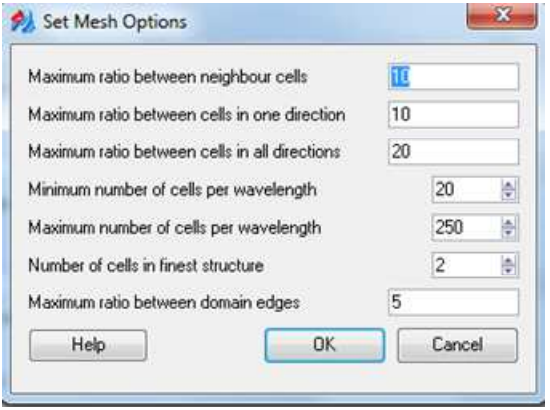
Loopup MinSize: Click to search for the minimum cell size.

Mesh： Generate the mesh distribution according to the problem structure and cell size.

Advanced Mesh: Set more mesh parameters.

Options： Open window to set more mesh parameters.

(3) Click the Options button to open the following window:



Maximum ratio between neighbor cells: Ratio between adjacent cell sizes.

Maximum ratio between cells in one direction: Ratio between the maximum and minimum cell sizes in one direction.

Maximum ratio between cells in all directions: Ratio between the maximum and minimum cell sizes in three directions.

Maximum number of cells per wavelength: Maximum number of cells per wavelength. The large number in this box is required to capture the fine structure during the mesh design.

Number of cells in finest structure: The number of cells in the fine structure. Two cells are required in the most cases to describe the narrow slot or thin dielectric layer.

Maximum ratio between domain edges: The ratio between the longest to shortest lengths.

After the mesh generation, follow the steps below to view the mesh distribution:

(1) Select the drawing plane, e.g. X-Y， Y-Z, or Z-X;

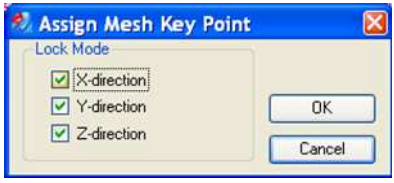
(2) Click on the corresponding button  ,  , or  ;

(3) Click on the button  in the toolbar, the mesh distribution appears. If the mesh distribution is embedded inside the model, you can adjust the height of drawing plane in the toolbar.

If you cannot generate the mesh that pass the special points in the model, you can improve the mesh distribution using the following way:

(1) Draw a point at the desired point. If the distance of two pints in one direction is smaller than the minimum cell size in the same direction, the center position of the two points will be key point.

(2) Draw a point at the desired point and assign it to be key point by clicking the icon  in the Mesh Accessory box. The following dialogue allows you to specify the key point in the selected directions:



You can select one direction or three directions, and force the mesh to pass those points.

(3) If you like to generate the mesh for a special object, you need to select the object and press the keys "Ctrl + L", and then "Ctrl + v". The object minimum frame will be line up with the mesh. You can select one surface of an object by clicking the  icon in the **Tool Box** for the same purpose.

(4) If you like to generate the mesh for a special object, you need to select the object and press the keys "Ctrl + k", and then "Ctrl + v". The detailed vertexes will be key points. You can select one surface of an object for the same purpose. You can select one surface of an object by clicking the  icon in the **Tool Box** for the same purpose.

Advanced Mesh is designed for the advanced users, and its most features will be integrated into the adaptive options.

7.5 Output Parameter Specification

The direct outputs include the near field, far field and frequently used derived parameters. The near field outputs include:

(1) Time domain field at the specified points.
(2) Field distribution on the specified lines in both the time domain and frequency domains.
(3) 2-D field distribution on the specified planes in both the time domain and frequency domains.
(4) 2-D SAR distribution on the specified planes in the frequency domain.
(5) 3-D SAR distribution in the specified volumes in the frequency domain.
(6) 1-g, 10-g and local peak SAR.
(7) Current in time domain.
(8) 3-D surface current distribution in the specified volumes.
(9) 3-D energy thermal distribution in the specified volumes.
(10) Voltage in the time domain.
(11) Mode current in the time domain.
(12) Mode voltage in the time domain.
(13) Mode current variation in the time domain.

(14) Mode voltage variation in the time domain.

(15) Mode propagation constant and attenuation factor.

(16) Total incident power (integral of E . J inside the source region).

(17) Forward power (Incident power – the dissipated power on the internal resistance).

(18) Accepted power (The power passes the input port = 1 – S11^2).

(19) Power density of a plane wave in the time domain.

(20) S-parameter and return loss

(21) S-parameter matrix for multiple port network

Far field outputs include:

(1) 2-D and 3-D Far field patterns.

(2) Polarization sense pattern.

(3) Axial ratio pattern.

(4) Tilt angle pattern.

(5) Radiation power.

(6) Monostatic RCS.

(7) Bistatic RCS.

(8) Directivity pattern.

(9) Gain pattern.

(10) Reflection and transmission coefficients of periodical structure.

Follow the steps given below to specify an output:

(1) Draw a 1-D, 2-D or 3-D geometry inside the domain;

(2) Select this object in the **Object Browser** box and assign an output option to it by clicking the output icon in the **Output** box or **Simulation Accessory** box of **Tool Box**.

(3) Change its name according to the output meaning in the **Property** box;

(4) The output option will be displayed in the **E/O** box in the **Properties** box.

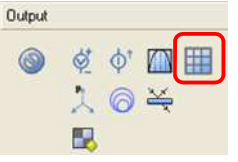
There are several windows for the output parameter specification.

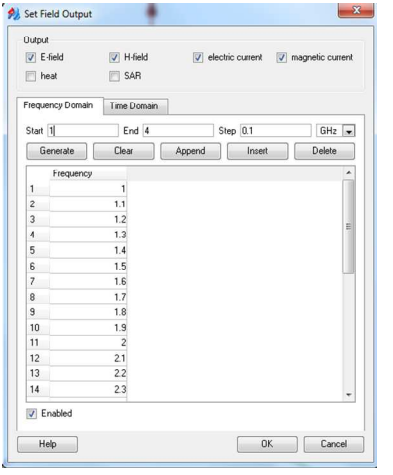
(1) Field outputs

(i) Electric and magnetic field output at a specified point, along a line, or on a specified surface;

(ii) Current density distribution on a specified surface;

(iii) SAR and thermal distribution.

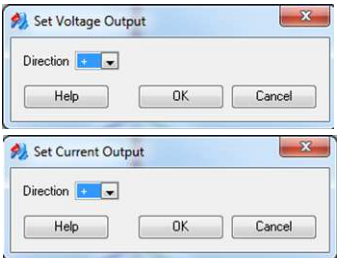




(2) Voltage and current outputs

(i) Time domain voltage output;

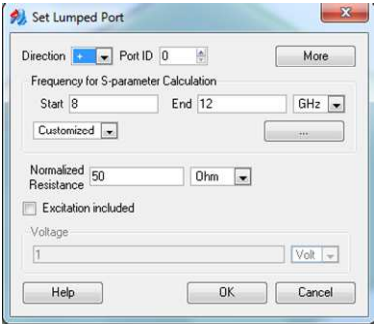
(ii) Time domain current output.



(3) Port outputs

(i) Lumped port outputs: s-parameters/return loss, time domain port voltage and current;

(ii) Wave port outputs: s-parameters/return loss, time domain port voltage and current.



Direction: Black end touches the positive pole for the lumped port. The ling segment points the wave propagation direction for the wave port.

Frequency for S-parameter Calculation->Start: Initial frequency

Frequency for S-parameter Calculation->End: End frequency

Frequency for S-parameter Calculation->(unit): Frequency unit

Frequency for S-parameter Calculation->Customized): Self-defined number of frequency sampling.

Frequency for S-parameter Calculation->(Adaptive): Default number of frequency sampling.

Normalized Resistance: Internal resistance of the excitation source, S-parameter is normalized to this value.

(4) Mode output

(i) Mode port S-parameter output;

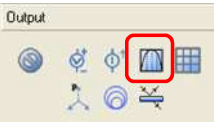
(ii) Mode port voltage;

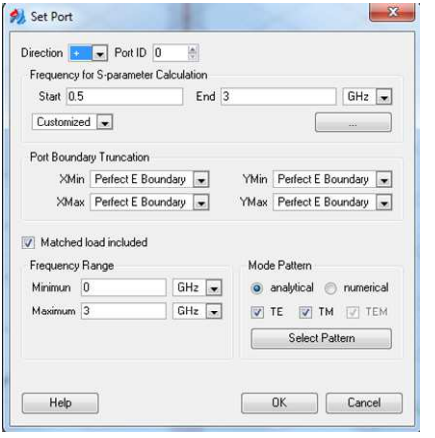
(iii) Mode port current;

(iv) Propagation constant;

(v) Attenuation factor;

(vi) Mode port impedance.





Direction: The line with a black end points to the wave propagation direction.

Frequency for S-parameter Calculation->Start: Initial frequency

Frequency for S-parameter Calculation->End: End frequency

Frequency for S-parameter Calculation->(unit): Frequency unit

Frequency for S-parameter Calculation->Customized): Self-defined number of frequency sampling.

Frequency for S-parameter Calculation->(Adaptive): Default number of frequency sampling.

Port boundary truncation->XMin: Boundary at the x minimum direction.

Port boundary truncation->XMax: Boundary at the x maximum direction.

Port boundary truncation->YMin: Boundary at the y minimum direction.

Port boundary truncation->YMax: Boundary at the y maximum direction.

Port boundary truncation->ZMin: Boundary at the z minimum direction.

Port boundary truncation->ZMax: Boundary at the z maximum direction.

For the symmetric structure, the proper boundary is selected in this window.

Matched load included: Check it for the open mode port.

Frequency range->Minimum: Pre-set initial frequency

Frequency range->Maximum: Pre-set end frequency

One port structure may support many modes, you can be only interested in the modes inside the frequency band of interest.

Mode Pattern->Analytic: For the rectangular and circular waveguide port, check this option will speed up the mode extraction.

Mode Pattern->Numerical: All other port structures, you need to check the **Numerical** button to get the correct results.

Mode Pattern->TE: Mode of interest is TE

Mode Pattern->TM: Mode of interest is TM

Mode Pattern->TEM: Mode of interest is TEM

Mode Pattern->Select Pattern: open the mode extraction window.

Click the **More** button to set more parameters.

(5) Far field

(i) 2-D far field pattern;

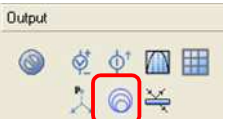
(ii) 3-D far field pattern;

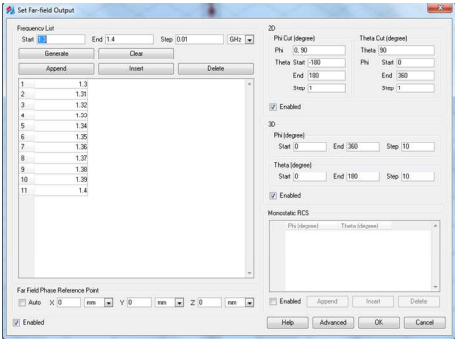
(iii) Monostatic RCS;

(iv) Bistatic RCS;

(v) Co-polarization and cross-polarization;

(vi) Equivalent current distribution on the Huygens' surfaces.

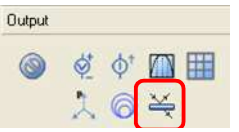


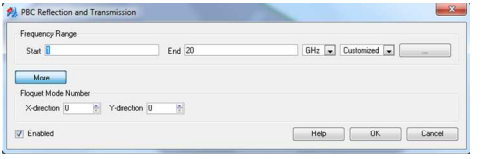


(6) FSS Reflection and Transmission

(i) Reflection coefficient output;

(ii) Transmission coefficient output.





Frequency range->Start: Start frequency of output parameters.

Frequency range->End: End frequency of output parameters.

Frequency range->Customized: User defined list of frequency sampling.

Frequency range->Adaptive: Default list of frequency sampling.

Click on the **More** button to set more output parameters.

Floquet mode number->X-direction: Floquet mode in the x-direction.

Floquet mode number->X-direction: Floquet mode in the x-direction.

(7) Surface current Output

(i) Select the metal objects inside the domain;
(ii) Click the **Surf_current** box in the **Properties** box;
(iii) Check the **Enabled** box;
(iv) Specify the output parameters for the 3-D surface current.

Maximum mesh length of surface: "-1": GEMS decides the minimum mesh size for surface current distribution. Any other number shows the specified maximum mesh length for the surface current distribution.

Frequency range->Start: Start frequency of output parameters.

Frequency range->End: End frequency of output parameters.

Frequency range->Step: Step size of frequency sampling.

Generate: Generate the frequency sampling list.

Enabled: Check the Enabled box to activate the surface output option.

7.7 Simulation of Dispersive Medium

In a dispersive medium, the material parameters such as permittivity or/and permeability are functions of frequencies. GEMS provides the dispersive medium types such as Debye, Lorentz, Plasma or known parameters at the discrete frequencies.

7.7.1 Debye Material

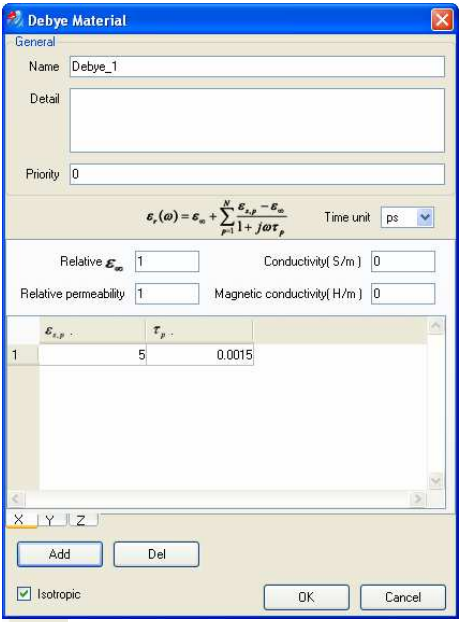
Follow the steps below to set the Debye material:

(1) Draw a 3-D object;

(2) Select this object in the **Object Browser** box;

(3) Click on the **Material** in the **Property** box;:

(4) Click on the **Debye** button in the **Material** window.

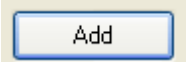


$\varepsilon_{s,p}$ Relative permittivity.

$\varepsilon_{s,p}$ Relative permittivity at zero frequency.

ε_{∞} Relative permittivity as the frequency approaches infinity.

τ_p Relaxation time.



Add one more pole

Check it for the material is isotropic.

7.7.2 Lorentz Material

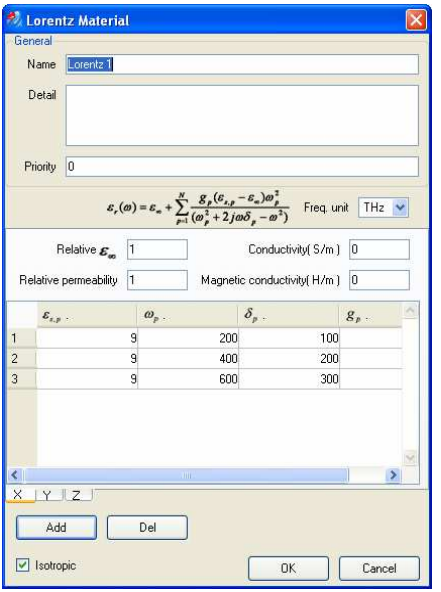
Follow the steps below to set the **Lorentz** material:

(1) Draw a 3-D object;

(2) Select this object in the **Object Browser** box;

(3) Click on the **Material** in the **Property** box;

(4) Click on the **Lorentz** button in the **Material** window.



$\varepsilon_{s,p}$ Relative permittivity at zero frequency.

ε_∞ Relative permittivity as the frequency approaches infinity.

g_p Weighted coefficient.

δ_p Dummy coefficient.

ω_p Resonant frequency of the medium.

Isotropic Check it for the material is isotropic.

7.7.3 Cold Plasma Material

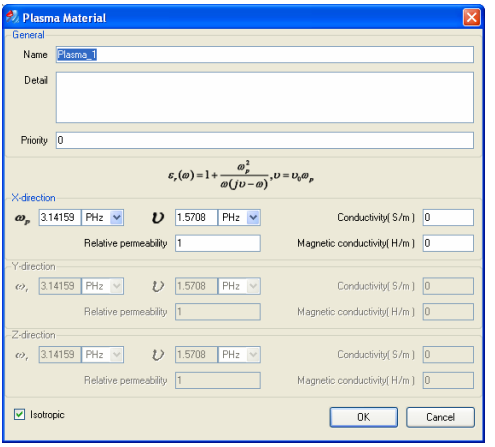
Follow the steps below to set the **Cold Plasma** material:

(1) Draw a 3-D object;

(2) Select this object in the **Object Browser** box;

(3) Click on the **Material** in the **Property** box;

(4) Click on the **Cold Plasma** button in the **Material** window.



υ Damping frequency.

ω_p Resonant frequency of the medium.

Isotropic Check it for the material is isotropic.

7.7.4 Drude Material

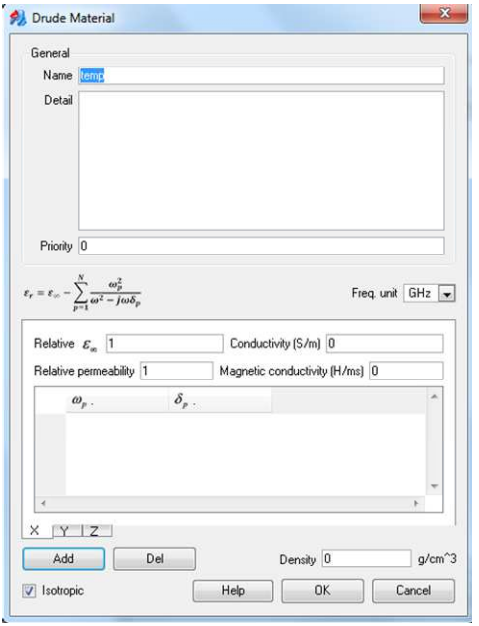
Follow the steps below to set the **Drude** material:

(1) Draw a 3-D object;

(2) Select this object in the **Object Browser** box;

(3) Click on the **Material** in the **Property** box;

(4) Click on the **Drude** button in the **Material** window.



ω_p Pole frequency of Drude model.

δ_p Inverse of relaxation time.

Check it for the material is isotropic.

7.7.5 Parametric Material

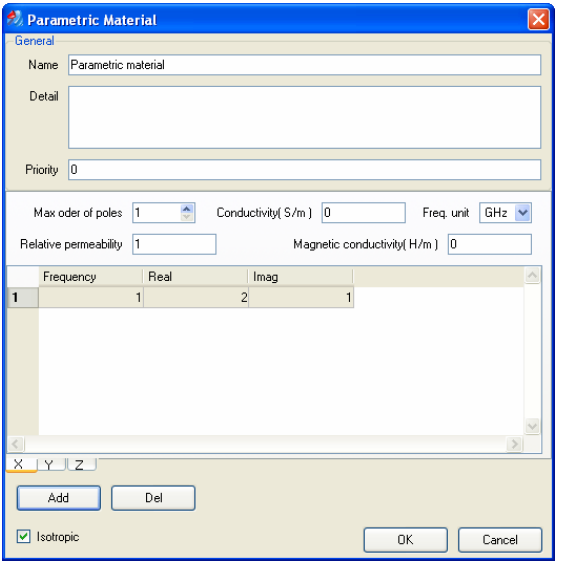
Follow the steps below to set the **Parametric** material:

(1) Draw a 3-D object;

(2) Select this object in the **Object Browser** box;

(3) Click on the **Material** in the **Property** box;

(4) Click on the **Parametric** button in the **Material** window.



If the dispersive relationship cannot be explicitly expressed in the formats described above, GEMS provides an interpolation module that will fit the sampling data using an approximation scheme.

7.8 Parallel Processing Setting

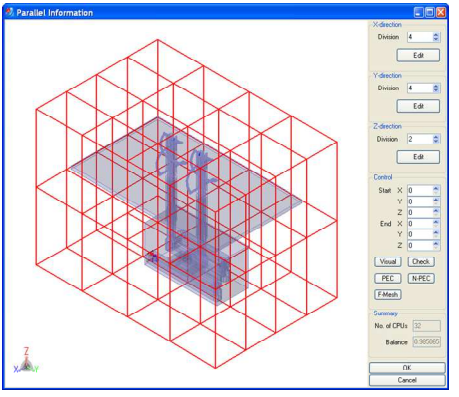
After you finish the structure input, mesh design, excitation and output setting, and boundary specification, follow the steps below to design the parallel processing:

(1) Click on the button in the toolbar;

(2) Specify the number of cores/CPUs/Nodes in each direction;

We recommend that you split the domain into subdomains based on the number of CPU or nodes to reach the better parallel processing performance. Some of advanced AMD CPUs includes two CPUs inside each physical CPU unit. You can handle them as two CPUs during the parallel processing.

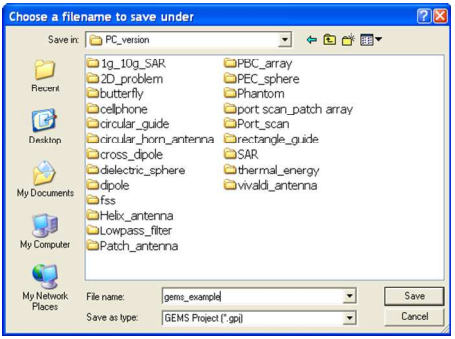
(3) Click on the **OK** button to finish the parallel processing design.



Though there are many advanced options in this window, however, they are designed for the advanced users. In most cases, you do not need to modify the default settings in this window.

7.9 Save GEMS Project

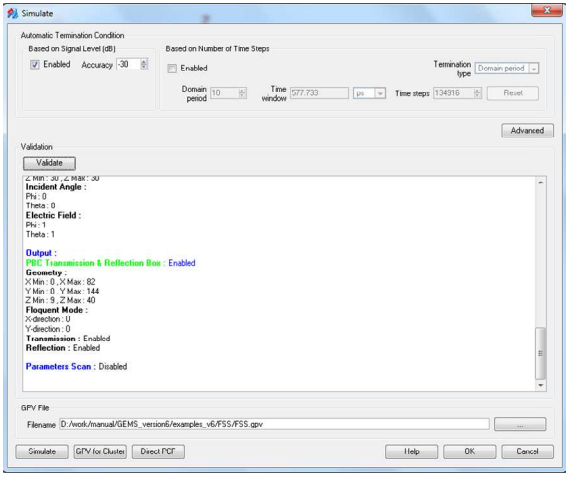
After finishing the project setting, click on the button to save the GEMS project.



The GEMS project file "*.gpj" includes the project model, boundary, excitation and output information, which can be edited by the GEMS designer.

7.9 Generating Project File

After you save the project file, click on the button  in the toolbar and specify the simulation parameters in the window below:



The options in the **Automatic Termination Condition** box:

Based on Signal Level->Enabled: Check it to activate the automatic termination condition.

Based on Signal Level->Accuracy: Convergent criteria in dB.

Based on Number of Time Steps->Enabled: Check it to activate the termination condition in which the simulation number of time steps is pre-set.

Based on Number of Time Steps->Termination type: Convergent type list.

Based on Number of Time Steps->Domain period: Use the number of time period that the wave propagates along the longest domain edge.

Based on Number of Time Steps->Time window: Use the absolute time period as the convergent criteria.

Based on Number of Time Steps->Time steps: Use the number of time steps.

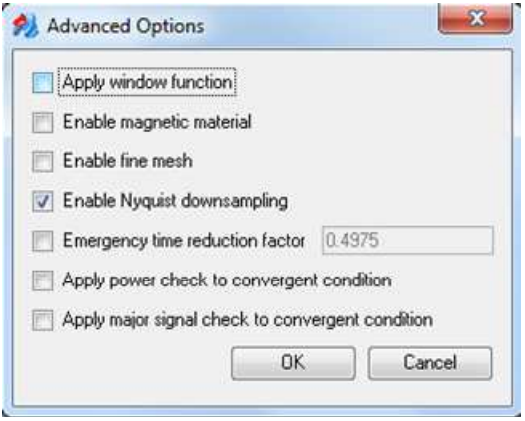
Based on Number of Time Steps->Reset: Reset the convergent criteria.

Simulate: Click on it to open the simulation window.

GPV for cluster: Generate *.gpv file that will be uploaded to cluster for simulation.

Direct PCF: Reserved

Click on the **Advanced** button to set up the advanced simulation option:



Apply window function: Add the Hamming function in the time domain signals. This option is only valid for the simulation with the fixed number of time steps.

Enable magnetic material: If the domain contains the magnetic material, this option is served as a switch. Uncheck it to ignore the magnetic material in the simulation.

Enable fine mesh: If the domain contains the subgridding region, this option is served as a switch. Uncheck it to ignore the subgridding region in the simulation.

Enable Niquist downsampling: Apply Niquist theorem in the DFT in the simulation.

Emergency time reduction factor: You can reduce the size of time step if necessary.

Apply power check to convergent condition: Add the power conservation in the convergent list.

Apply major signal check to convergent condition: For the weak coupling problems, apply the special convergent criteria in the simulation.

For GEMS PC version, follow the steps below generate simulation project file:

(1) Select convergent criteria;

(2) Select the advanced options if necessary by clicking the **Advanced** button;

(3) Click on the **Validate** button to check if the project includes the invalid options (in red color);

(4) Check if the simulation file, *.gpv, is in the correct folder;

(5) Click on the **Simulate** button to open the simulation window.

For GEMS Linux version, follow the steps below to generate the simulation project file:

(1) Select convergent criteria;

(2) Select the advanced options if necessary by clicking the **Advanced** button;

(3) Click on the **Validate** button to check if the project includes the invalid options (in red color);

(4) Check if the simulation file, *.gpv, is in the correct folder;

(5) Click on the **GPV for cluster** button to generate the *.gpv file;

(6) Click on the **OK** button in the popup window if *.gpv file is generated successfully;

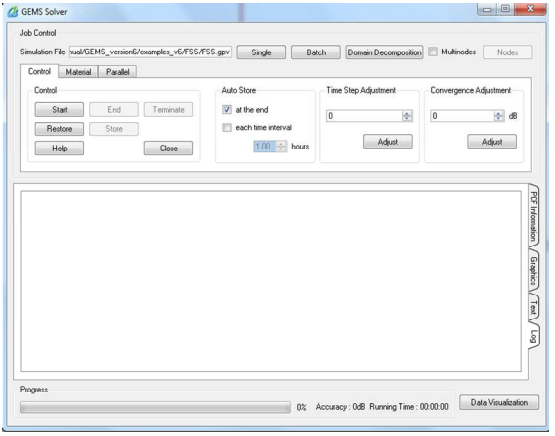
(7) Upload *.gpv file or its zipped file if it is big to the cluster through GEMS WBI (or other software such as SSH).

Chapter 8 GEMS Project Simulation

GEMS software can be run on PC, workstation, PC cluster, and high performance cluster installed with both Windows and Linux operating system.

8.1 Run GEMS Project on PC

You can start the simulation by clicking the **Simulate** button in the **Project File Generation** window, or, select the option solver in the GEMS folder of **Start** program to launch the GEMS simulation window.



If starting the simulation by clicking the **Simulate** button in the **Project File Generation** window, the ***.gpv** simulation file will be loaded into the GEMS solver window automatically.
If the **Start** button is not active, you need to click on the **Single** button to reload the *.gpv file. Click on the **Data Visualization** button to view and process the simulation results after the simulation is completed.

8.1.1 Button and Option Explanations

We now explain the buttons and options in the GEMS Solver window.

(1) **Simulation file**: List the project path and name that you like to simulate.

(2) **Single**: Select a single project for simulation.

(3) **Batch**: Select multiple projects for simulation. GEMS will simulate them once at one time in the order in the list.

(4) **Domain Decomposition**: Divide the domain into small pieces for the fast project preprocessing. Windows system maximally allows you to break the domain into several hundreds of pierces.

(5) **MultiNodes**: Check it to use the multiple computers for simulation.

(6) **Control** box

Control->Start: Click on it to start the simulation.

Control->End: Terminate the current project simulation and you will have the complete frequency domain results.

Control->Terminate: Terminate the current project simulation and you may loss the frequency domain results.

Control->Store: Store the intermediate results for the continue simulation.

Control->Restore: Resume the simulation based on the previous results if you properly store the intermediate results before.

Control->Help: Open the help document.

Control->Close: Close the GEMS solver window and stop the simulation.

Auto Store->At end: Save the intermediate results after the simulation so that you can resume the simulation next time.

Auto Store->Each time interval: Specify a time period to save the intermediate results after the simulation so that you can resume the simulation from the last broken point. The unit is hour.

Time Step Adjust: You can extend the number of time steps during the simulation for the fix time step simulation. Type a proper number in the box and then click on the **Adjust** button. This option is not good for the simulation with window function option.

Convergence Adjust: Adjust the convergent criteria during the simulation. Type a proper number in the box and then click on the **Adjust** button. This option is not good for the simulation with window function option.

(7) **Material box**

Conformal Setting->Type: Options for the conformal levels.

Standard: the conformal option does not need to reduce the size of time step.

Advances: the conformal option may require to use the small size of time step.

Conformal Setting->Factor: control the level of the conformal technique. The small factor will reduce the size of time step.

Material Output->Conductivity: Check to output the conductivity distribution inside the computational domain to help check if the model is correct.

(8) Parallel box

Processing Optimization->PEC conformal: If the domain includes the complex PEC structure and the project pre-processing is slow, check this option to speed up the project pre-processing.

Processing Optimization->Dielectric Conformal: If the domain includes the complex dielectric structure and the project pre-processing is slow, check this option to speed up the project pre-processing.

Two options above may require extra memory than usual.

Runtime Parallelization->Auto: If you do not specify the core distribution in the GEMS designer, check this box and GEMS will design the parallelization automatically according to the number of cores in your computer. For example, the computer has 8 cores, the core distribution may be 8×1×1, 4×2×1, 2×2×2, etc. depending on the domain shape. If you do not do any parallelization design in the designer and do not check the **Auto** box neither in the GEMS solver, GEMS will automatically use the OpenMP option to use all the cores in the GEMS simulation.

8.1.2 Simulation Status

The simulation status box has four labels:

(1) **PCF Information**: Display the information about the parallel processing such as the number of processes, total number of cells in each direction, number of processes in each direction, and number of cells in each direction in each process.

(2) **Graphics**: Display the simulation progress graphically.

(3) **Text**: Display the simulation progress in the text format.

(4) **Log**: Display the simulation log that shows you more simulation status including the simulation time for each part.

8.1.3 Simulation Progress

During the simulation, you can monitor the simulation progress.

(1) The simulation progress is displayed in the progress bar, the convergent level and simulation time are also shown at the bottom of the GEMS silver window.

(2) Click the **Data Visualization** button to open the GEMS Display window and view the intermediate simulation results.

8.2 Run GEMS Project on PC Cluster

Similar to the procedure on a PC, for the cluster simulation, you can also start the simulation by clicking the **Simulate** button in the **Project File Generation** window, or, select the option 

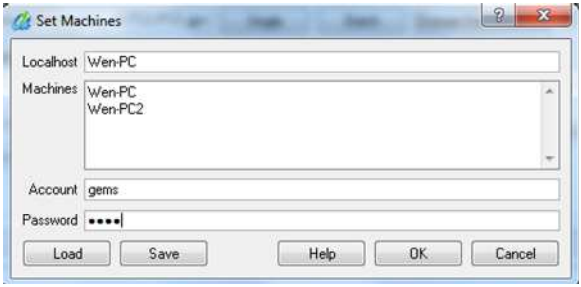
in the GEMS folder of **Start** program to launch the GEMS simulation window.

If a PC cluster is installed with Windows system, you can submit the GEMS job from any computer in the cluster.

Besides the same steps used in the PC simulation, for the PC cluster simulation, you need to do two more things below:

(1) Check the **MiltiNodes** button;

(2) Click on the **Nodes** button to list computer names or IPs in the cluster. The current computer name or IP should be at the first place in the list. You can assign one sub-domain to one node/computer, one CPU, or one core. You can specify the number of cores for simulation in the following format:



Localhost: GEMS will check and load the current computer name in this box.

Machines: Computer name or IP address list. The current computer name must at the first place.

The user must have an account on each computer with the same username and password,

Load: If you have stored the machine list of computer names in a text file (*.lst), you can load it without typing the computer name each time.

Save: Click to save the list of computer names to a text file (*.lst).

Help: Open the help document.

Ok: Confirm the list and account information, and close the window.

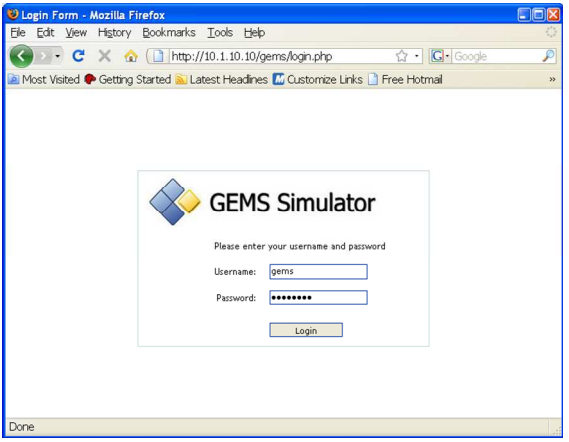
Cancel: Cancel the setting.

8.3 Run GEMS Project Using WBI

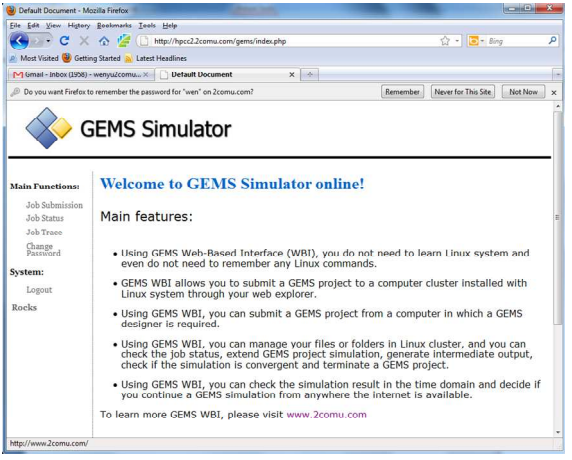
In this part, we guide you how to run a GEMS project in a Linux platform through the GEMS web based interface (WBI). The GEMS WBI requires that your cluster has been installed with either PBS or SGE management software.

To run the GEMS jobs on a Linux cluster, you need to have a valid account and the GEMS package has been properly installed on the cluster. In order to log in the cluster, you need to open the Firefox or IE web browser, and type the IP or

internet address of the cluster in the address textbox. The following page will appear and you can enter the username and password in the username and password boxes, respectively.



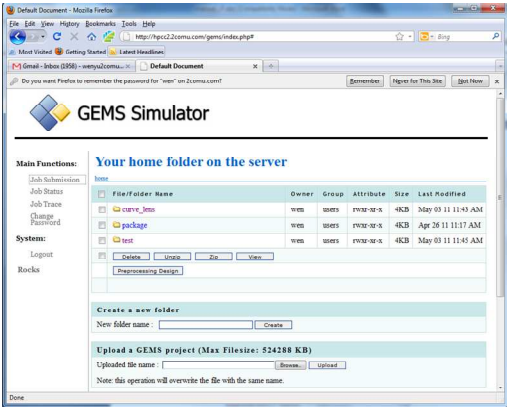
If the username and password have been accepted, the following page will show up. Using GEMS WBI, you can submit a GMES job through web browser (Firefox or IE), check job status, check the simulation progress (limited to some output options), change your password, and check the cluster status if the cluster is installed with Racks software.



Click on the *Job Submission* link at the left of the window, your files and folders in this cluster will be listed in the following window. The listed file or folder properties include the Owner, Group, Attributes and Last Modified date.

- **Delete**: Delete the selected files or folders.
- **Unzip**: Unzip a zipped file. To speed up the load speed, you should upload a zipped project file, and unzip it in WBI window.
- **Zip**: Zip a file or folder. To speed up the load speed, you should zip a folder or file before you download a folder or file.
- **View**: View the selected file.
- **Pre-Processing Design**: Load the project file and open the pre-processing design window.
- Click on a file to download it.
- **New folder name**: Type a folder name in the **New Folder Name** box and then click the **Create** button to create a folder on the cluster.
- **Upload File Name**: Click on the Browser button to select a project file on your local computer, and then click on the Upload button to upload it to the cluster.

Create a new folder by typing the folder name in the box labeled with "**New folder name**". The existing folder with the same name will be overwritten.



.

Check the checkbox of the file ***.gpv**, and click on the **Pre-processing Design** button. The numbers of subdomains does not have to equal to the number or compute cores or processors, and it can be any number but their multiplication should be smaller than 1024. The default simulation time is 24 hours, namely, the job will be killed the simulation time reaches 24 hours regardless of the simulation status. You can modify it if the job will take longer than 24 hours.

- Show the number of processors required by the current project.

- Start from beginning: check it to start the simulation as a new project.
- Resume from last store: Check it to start a simulation from the last time if you properly save the intermediate results in the last simulation.
- Automatically store at end: Save the intermediate results after the simulation.
- Automatically store for each hours: Save the intermediate results every time period specified in the text box.

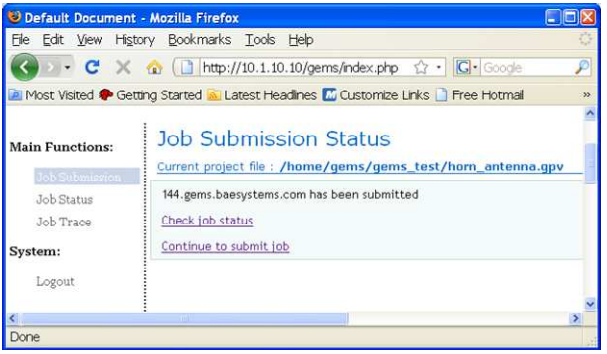
- Number of grids in the X-direction: show the total number of cells in the x-direction.
- Number of grids in the Y-direction: show the total number of cells in the y-direction.
- Number of grids in the Z-direction: show the total number of cells in the z-direction.
- Partition number of domain decomposition: number of sub-domains in the x-direction for the project pre-processing.
- Partition number of domain decomposition: number of sub-domains in the y-direction for the project pre-processing.
- Partition number of domain decomposition: number of sub-domains in the z-direction for the project pre-processing.

- Conductivity distribution inside computational domain: check it to output the conductivity distribution inside the computational domain.

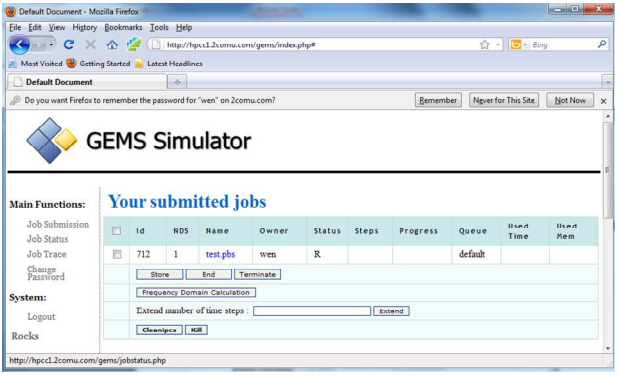
- Estimate simulation hours: GEMS simulation can last the maximum time regardless of the simulation status.

- Conformal Type: **Standard**: the conformal option does not need to reduce the size of time step. **Advances**: the conformal option may require to use the small size of time step.

Click on the **Submit** button to submit the selected job to the cluster for the simulation. If the job has been successfully submitted, you will see the following message.

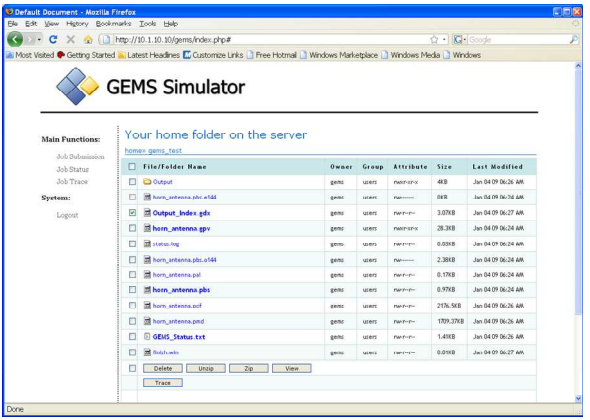


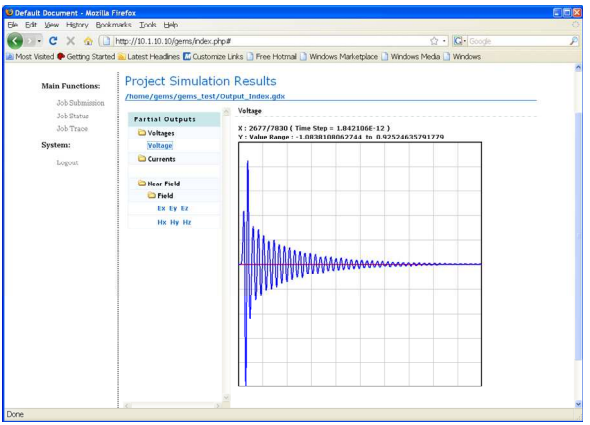
Click on the **Check job** status to check the job status.



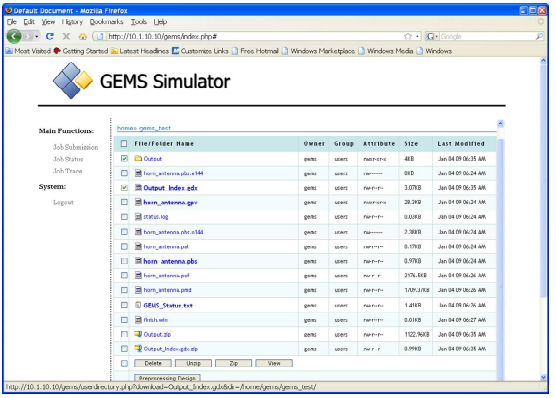
GEMS will update the status every minute, however, you can click on the **Job Status** link to update the simulation status at any time.

If the GEMS project includes the output such as voltage, current, port, or field at point, you can check the time signature graphically. To this end, go to the current folder, and check the ***.gdx** file and then click on the **Trace** button to view the time signature.





The simulation time and log are in the *.oxxx. GEMS allows you to terminate the simulation or extend the simulation time steps if necessary. After the job is completed, you can check the **Output** folder and ***.gdx** file, and then click on the zip button to zip the result into a zipped file. Click on the zipped file to download it to your local PC for the result visualization and data post-processing.



Unzip the result file on your local PC for the result visualization and data post-processing through the GEMS display module.

GEMS allows the administrator to create a new account following the steps below:

(1) Login the cluster using the root information;

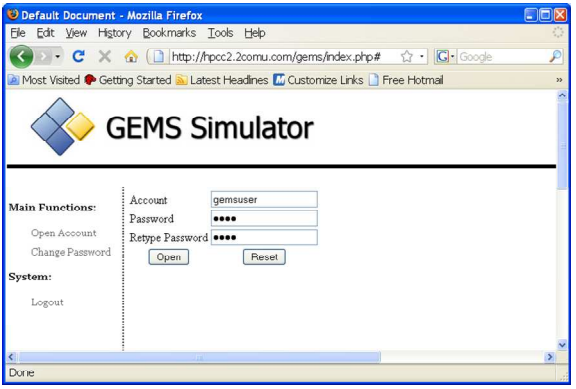
(2) Click on the **Open Account** link;

(3) Input the username you will create;

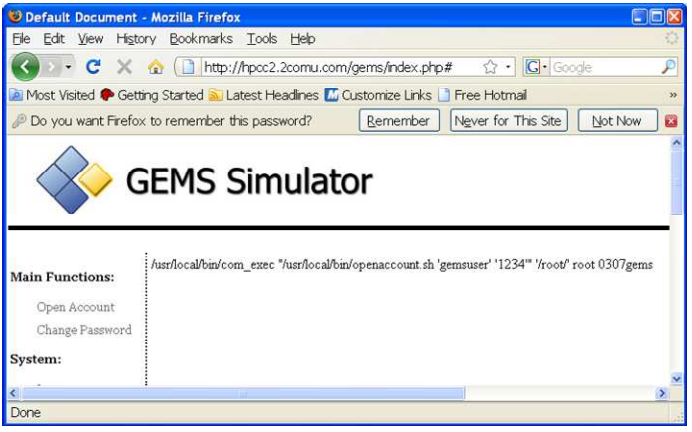
(4) Input the password;

(5) Reinput the password;

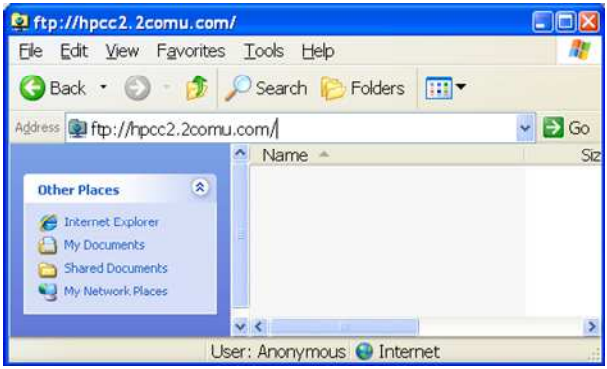
(6) Click on the **Open** button to create a new account.



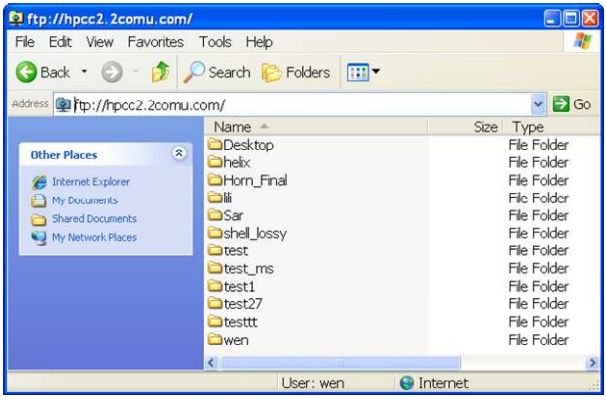
GEMS will return a message as follows if the new account is successfully created. In the current GEMS WBI version, you are not allowed to remove an existing account.



GEMS also provides a FTP service, which allows you easily upload and download files and folders between your PC and the cluster. Type the name or IP address of your cluster in your local file browser, for example, "ftp://hpcc2.2comu.com", and type your name and password in the pop-up window.



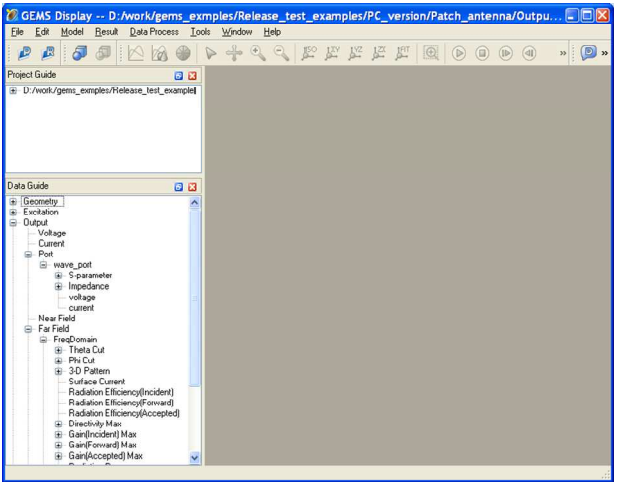
Your files and folders on the cluster will appear in your file browser, and you can handle the files and folders as in the local computer.



Chapter 9 GEMS Display Module

GEMS display includes three major parts: the direct result visualization module and the data post-processing module. Selecting the option 

in the **GEMS** folder of the **Start** program to launch the GEMS display module.



GEMS display window includes the following major buttons in its toolbar:


Open a GEMS project; open the corresponding output index file. If you click on the first button, GEMS will automatically open both the ***.gpv** and ***.gdx** files.


View the project model; view the project model in the transparent style.

 Plot the selected result in the result tree in a new figure in Cartesian coordinate system; plot the result on an existing figure; plot the far field pattern in a new polar coordinate system.

 Deselect a select; Read the value at the selected point; Zoom in the view region; Zoom out the view region.

 View a project model or a 3-D result in different formats: reset to ISO; view the project model in the x-y plane; y-z plane; z-x plane and fit to the view window.

 Start 2-D result animation, or, plot the 3-D data with the model: stop the animation; forward one step; backward one step; sampling step; and display rate.

 Open the data post-processing center.

 Open the time domain signature extrapolation window.

Plot a result in the Smith Chart (in the **Result** menu).

The project model can be displayed independently in either the transparent or the regular format. To view the project model, you need to load the project and result index files first, and then click on the  button to view the project model. Click on the  button to view the project model in the transparent format.

9.1 Open Project for Result Display

To visualize GEMS project result by clicking on the  button and selecting the project file ***.gpv**. You can open more than one project in the same display window. The result tree includes the following major entrances:

(1) Geometry information (for view only).

(2) Excitation sources (including excitation pulse, incident field of plane wave and incident power information).

(3) Direct output results.

(4) Project mesh information.

(5) Customized outputs.

The **Refresh** option in the **File** menu will reload the opened project results. You can select and export a GEMS result into a data file, and you can also import some data files into the result tree.

GEMS display module includes a data post-processing center, which allows you to process the GEMS direct outputs to derive the new results.

9.2 GEMS outputs

GEMS direct outputs include:

(1) Time domain results

(a) Current.

(b) Voltage.

(c) Port voltage.

(d) Port current.

(e) Field at specified point.

(f) Field distribution on a specified line.

(g) Filed distribution on the specified surfaces.

(h) Current density distribution on the specified surfaces.

(i) Mode current and voltage.

(j) Total incident power.

(k) Field density of plane wave.

(l) Far field at specified angles.

(2) Frequency domain results

(a) Port S-parameters or return loss.

(b) S-parameter matrix for multiple port network.

(c) Port impedance.

(d) Field distribution on a line at the specified frequencies.

(e) Field distribution on a surface at the specified frequencies.

(f) Current density distribution on the specified surfaces and frequencies.

(g) 2-D far field pattern (directivity, gain, polarization, axial ratio, and tilt angle).

(h) 3-D far field pattern.

(i) Total active reflection coefficient.

(j) 3-D surface current density distribution.

(k) Radiation power.

(l) Accepted power (calculate from S-parameter).

(m) Forward power (total incident power – Source dissipation power).

(n) Source dissipation power.

(o) Total power (Integral of E and J in the excitation source region).

(p) Radiation efficiency normalized to the three powers.

(q) 3-D thermal distribution.

(r) 2-D and 3-D SAR distributions at the specified frequencies.

(s) 1-g, 10-g and local peak SAR.

(t) Maximum SAR and averaging SAR inside a specified region.

9.3 Basic Steps to Plot 1-D Results

Plot a curve in the Cartesian coordinate system.

(1) Select the result option in the result tree;

(2) Double-click on the option, or click on the button  in the toolbar to plot it in the Cartesian coordinate system.

Plot a curve in the polar coordinate system in the linear scale.

(1) Select the result option in the result tree

(2) Click on the  in the toolbar

Plot a return loss or s-parameter in the Smith Chart.

(1) Select the result option in the result tree;

(2) Select the **Smith Chart** option in the **Result** menu.

Plot multiple curves in one figure

(1) Plot the first curve in a new figure;

(2) Select the second option in the result tree;

(3) Clicking on the button  to plot the selected result onto an existing figure;

(4) Follow the same procedure to plot more curves in the same figure.

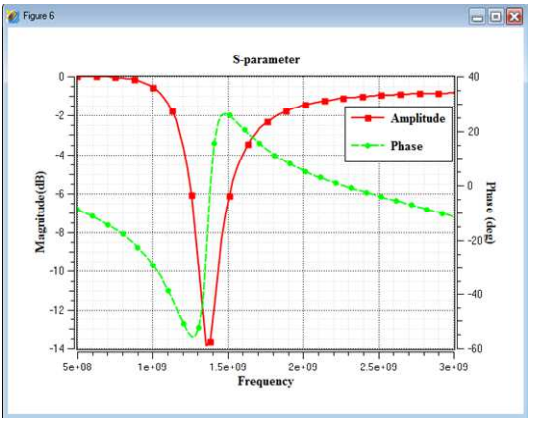
Control 2-D curve options in Cartesian coordinate system

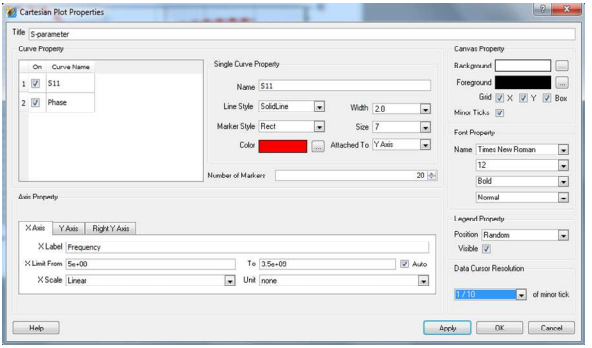
(1) Select an option in the result tree;

(2) Double click on it or click on the "View in New Window" button in the toolbar;

(3) Double-click on anywhere inside the figure window;

(4) Modify the figure options in the window.





- **Title**: you can edit the figure title
- **Curve Property->On**: Checked curve will be displayed in the figure window. Unchecked curve will be hidden in the figure window.
- **Curve Property->Curve name**: show the curve name, and you can select and edit it in the "Curve Property->Single Curve Property->Name" box.

- **Curve Property->Single Curve Property->Name**: Show the name of the selected curve.
- **Curve Property->Single Curve Property->Line Style**: Line style list.
- **Curve Property->Single Curve Property->Width**: Line width control.
- **Curve Property->Single Curve Property->Marker Style**: Marker style for the selected curve.
- **Curve Property->Single Curve Property->Size**: Marker size for the selected curve.
- **Curve Property->Single Curve Property->Color**: Color for the selected curve.
- **Curve Property->Single Curve Property->Attached To**: Vertical axis (left or right axis) associated to the selected curve.
- **Curve Property->Single Curve Property->Number of Markers**: Control number of markers in the curve.

- **Axis Property->X Axis->X Label**: Show the label of axis.
- **Axis Property->X Axis->X Limit From**: Display range of the axis. Check the Auto box to reset the display range to the default.
- **Axis Property->X Axis->X Scale**: Display scale (Linear or Log10) of vertical axis.
- **Axis Property->X Axis->Unit**: "none" means the default including in the original data.

- **Canvas Property->Background**: Show the background color of the figure windows.

- **Canvas Property->Foreground**: Show the curve color in the figure windows.
- **Canvas Property->Grid->X**: Check to show the grids in the x-direction.
- **Canvas Property->Grid->Y**: Check to show the grids in the y-direction.
- **Canvas Property->Grid->Box**: Check to show the box frame of the figure window.

- **Font Property->Name**: Show the font name.
- **Font Property->(Size)**: Show the font size.
- **Font Property->(Property)**: Show the font in the bold, light, normal, demi, or black.
- **Font Property->(Type)**: Show the font in the normal, italic or oblique.

- **Legend Property->Position**: Display the legend at the different places.
- **Legend Property->Visual**: Check to show the legend.

- **Data Cursor Resolution**: The resolution to read the value on the curves.

- **Apply**: To preview the selected options.
- **Ok**: Confirm the specified options.
- **Cancel**: cancel the specified options.
- **Help**: Show the help document.

Read the curve value following the steps below:

(1) Plot a 2-D option in Cartesian system;

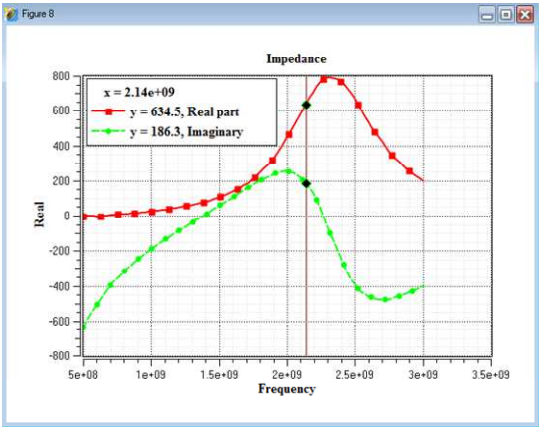
(2) Select another 2-D option and plot them in the same figure;

(3) Click on the **Data Cursor** button in the toolbar;

(4) Move the mouse icon to the figure region;

(5) Select the start point and press the left mouse button;

(6) Click on the **Select** button in the toolbar to deselect this function.



The angle and radius of selected point are shown in the legend box.

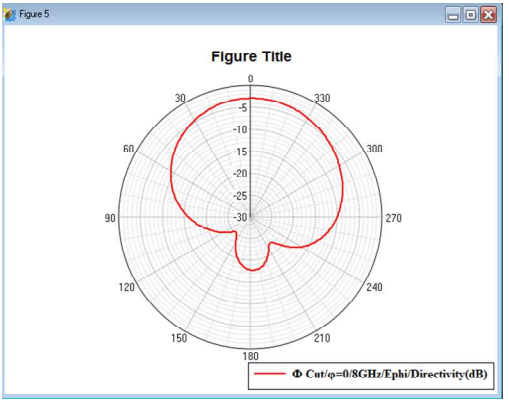
Control 2-D curve options in polar coordinate system

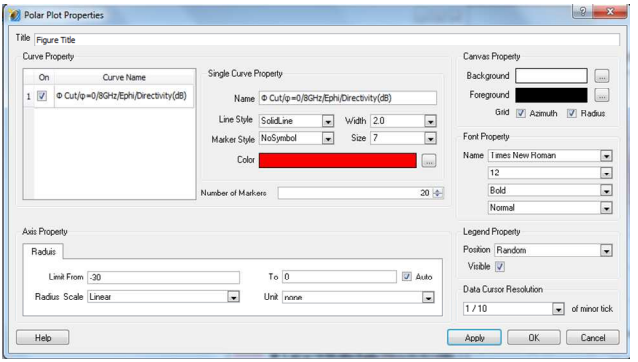
(1) Select an option in the result tree;

(2) Double click on it or click on the "View in Polar" button in the toolbar;

(3) Double-click on anywhere inside the figure window;

(4) Modify the figure options in the window.





Title: you can edit the figure title

- **Curve Property->On**: Checked curve will be displayed in the figure window. Unchecked curve will be hidden in the figure window.
- **Curve Property->Curve name**: show the curve name, and you can select and edit it in the "Curve Property->Single Curve Property->Name" box.
- **Curve Property->Single Curve Property->Name**: Show the name of the selected curve.
- **Curve Property->Single Curve Property->Line Style**: Line style list.
- **Curve Property->Single Curve Property->Width**: Line width control.
- **Curve Property->Single Curve Property->Marker Style**: Marker style for the selected curve.

- **Curve Property->Single Curve Property->Size**: Marker size for the selected curve.
- **Curve Property->Single Curve Property->Color**: Color for the selected curve.
- **Curve Property->Single Curve Property->Number of Markers**: Control number of markers in the curve.

- **Axis Property->Radius->Limit From**: Display radius range of the axis. Check the Auto box to reset the display range to the default.
- **Axis Property->Radius->Radius Scale**: Display scale (Linear or Log10) of vertical axis.

- **Canvas Property->Background**: Show the background color of the figure windows.
- **Canvas Property->Foreground**: Show the curve color in the figure windows.
- **Canvas Property->Grid->Azimuth**: Check to show the grids in the azimuthal direction.
- **Canvas Property->Grid->Radius**: Check to show the grids in the radius direction.

- **Font Property->Name**: Show the font name.
- **Font Property->(Size)**: Show the font size.
- **Font Property->(Property)**: Show the font in the bold, light, normal, demi, or black.
- **Font Property->(Type)**: Show the font in the normal, italic or oblique.

- **Legend Property->Position**: Display the legend at the different places.
- **Legend Property->Visual**: Check to show the legend.

- **Data Cursor Resolution**: The resolution to read the value on the curves.

- **Apply**: To preview the selected options.
- **Ok**: Confirm the specified options.
- **Cancel**: cancel the specified options.
- **Help**: Show the help document.

Read the curve value following the steps below:

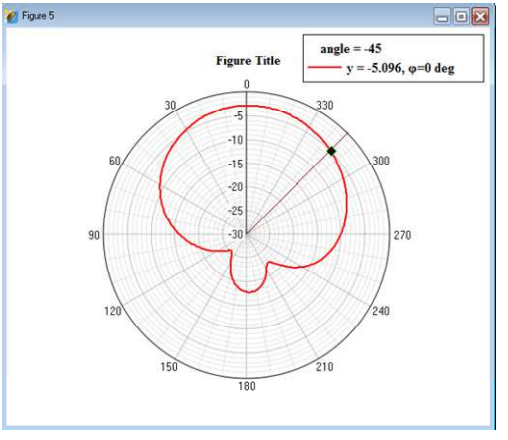
(1) Plot a far field pattern in polar system;

(2) Click on the **Data Cursor** button in the toolbar;

(3) Move the mouse icon to the figure region;

(4) Select the start point and press the left mouse button;

(5) Click on the **Select** button in the toolbar to deselect this function.

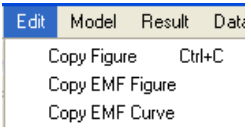


The angle and radius of selected point are shown in the legend box.

Export the figures

(1) Select one figure (active);

(2) Select the export format from the **Edit** menu to copy the selected figure to the clipboard;



Copy Figure: BMP format.

Copy EMF Figure: Vector format.

Copy EMF Curve: Curve only in the vector format.

The vector format figure allows you to edit it in the Microsoft PowerPoint or Words by double-clicking on it two times.

Export the result to a text file

(1) Select one option or multiple options that have the same attribute in the result tree

(2) Select the **Export** option in the **File** menu to export the result to a data file.

9.4 Basic Steps to Plot 2-D Results

(1) Select the result option in the result tree;

(2) Double-click on the option, or, click on the button in the toolbar to plot it in the project model;

(3) Change the display format using the options in Result->3D Control;

(4) Change the reference in the Tool->Options if necessary;

(5) For the time domain distributions, use the  options to animate the results;

(6) Select the **Export** option in the **File** menu to export it into a data file.

(7) Select the **Copy Figure** option in the **Edit** menu to copy the selected figure to the clipboard;

(8) There are more advanced operations on a 2-D figure in the 3D control in the **Result** menu, for example:

(i) Switch the dB and linear scales;

(ii) Contour plot;

(iii) View the cutting plane;

(iv) Pick up a curve from a 2-D result.

9.5 Basic Steps to Plot a 2-D Result with Project Model

(1) Click on the  button in the toolbar;

(2) Select the result option in the result tree;

(3) Double-click on the option, or, click on the button  to plot it in the project model;

(4) Change the reference in the Tool->Options if necessary;

(5) Change the display format using the options in Result->3D Control;

(6) For the time domain distributions, use the buttons  to animate the results;

(7) Select the **Export** option in the **File** menu to export it into a data file;

(8) Select the **Copy Figure** option in the **Edit** menu to copy the selected figure to the clipboard.

9.6 Basic Steps to Plot Surface Current Distributions

(1) Select the surface current distribution option in the result tree;

(2) Click on the button  to plot it in the project model;

(3) Click on the button  if the result is blocked;

(4) Change the reference in the Tool->Options if necessary;

(5) Change the display format using the options in Result->3D Control;

(6) Select the Copy Figure option in the Edit menu to copy the selected figure to the clipboard.

9.7 Basic Steps to Plot Surface Current Distributions with Project Model

(1) Click on the  button in the toolbar

(2) Select the surface current distribution option in the result tree;

(3) Click on the button  to plot it in the project model;

(4) Click on the button  if the result is blocked;

(5) Change the reference in the Tool->Options if necessary;

(6) Change the display format using the options in Result->3D Control;

(7) Select the **Copy Figure** option in the **Edit** menu to copy the selected figure to the clipboard.

9.8 Basic Steps to Plot a 3-D SAR or Thermal Distribution

(1) Select the 3-D SAR or thermal distribution option in the result tree;

(2) Change the reference in the Tool->Options if necessary;

(3) Click on the button  , or double-click on it to plot it;

(4) Change the display by using the options in the **Result** -> **3D Control** -> **Volume Options**.

(5) Select the **Copy Figure** option in the **Edit** menu to copy the selected figure to the clipboard.

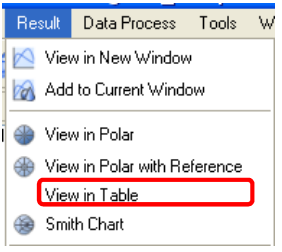
9.9 Basic Step to View the Maximum Power or Efficiency in Table

The basic steps to display the maximum values of radiation efficiency, directivity, gain, radiation power, 1-g SAR, 10-g SAR and local peak SAR in a table:

(1) Select one option in the result tree;

(2) Click on it to plot it (cannot display position of the maximum value);

(3) Select the **View in Table** option in the **Result** menu.



(4) Select the **Export** option in the **File** menu to export the result to a data file.

9.10 Export More Options into One Text File

GEMS allows you to export several options in the result tree into one data file if they have the same data type. Follow the steps below to export four options into one file:

(1) Select four options in the result tree;

(2) Select the **Export** option in the **File** menu;

(3) Specify a file name.

9.11 S-Parameter Matrix

GEMS can generate the s-parameter matrix of a multiple port network in a single simulation. For example, a 4-port network has a port 1, 2, 3, and 4, which are excited once a time. The 4 ports will be listed in the port option in the result tree. Under the port_1, the S_{11} is located in the "1" folder (the port 1 is excited); S_{12} is located in the "2" folder (the port 2 is excited); S_{13} is located in the "3" folder (the port 3 is excited); and, S_{14} is located in the "4" folder (the port 4 is excited). Follow the exactly same pattern for other ports.

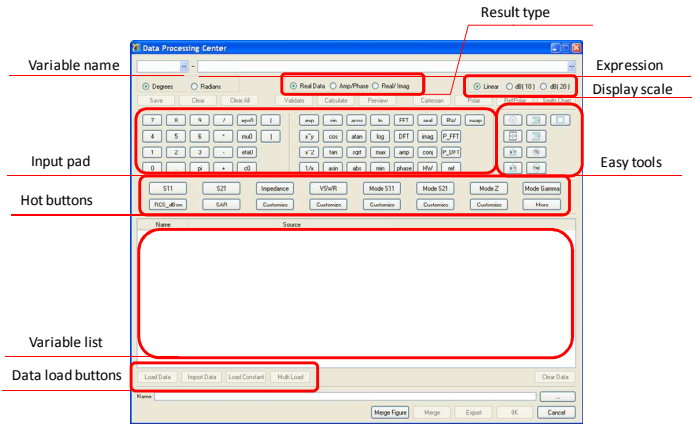
The complete matrix of 4-port network is stored in the Optimization->S Matrix folder in the TouchStone format. You can select it and then the **Export** option in the **File** menu to export it in the TochStone format.

Chapter 10 Derive Your Own Results

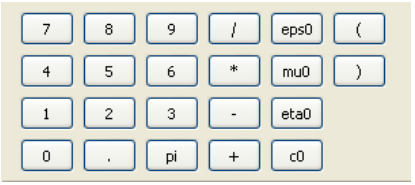
The data post-processing center helps you generate the output parameters from the basic outputs of the GEMS solver.

10.1 Introduction to Buttons and Functions

Besides the direct result listed in the result tree, GEMS also provides a powerful data post-processing module that can be used to help you derive different output parameters from the GEMS direct outputs. This module allows you to input the variables and expression, which are filled with the GEMS results, to generate the output parameters of interest. Click on the  button in the GEMS display window.



Regular calculator module



Where **eps0**, **mu0** and **eta0** are the permittivity, the permeability and the wave impedance of free space, respectively, and **c0** is the speed of light in free space.

GEMS calculator module

Where **amp** and **conj** stand for the amplitude and conjugate of a complex number/variable, respectively. Besides some commonly used functions in the panel above, GEMS defines its own five functions:

(a) **HW** is a Hamming window function. For example, a time signature $E(t)$ can be multiplied by the Hamming window function, and is expressed as $E(t)$ * **HW**, which is still a time signature.

(b) **BW** is used to extract the 3-dB beam width of a far field pattern.

(c) **P_DFT** is used to calculate the total incident power from excitation source to the system using the discrete Fourier transformation.

(d) **P_FFT** is used to calculate the total incident power from excitation source to the system using the fast Fourier transformation.

(e) **Ref** is a function to take the time index of a time signature, the frequencies of a frequency domain data. For example, **Ref**(**FFT**($E(t)$)) will output a frequency list, which corresponds to the frequencies of **FFT**($E(t)$).

GEMS special parameter module

Calculate the characteristic impedance of a circular coax, where the parameters **a** and **b** are the inner and outer radii of the circular coax.

Calculate the characteristic impedance of a rectangular coax, where the parameters $\mathbf{w_1}$ and $\mathbf{w_2}$ are the inner and outer sides of the rectangular coax.

Calculate the characteristic impedance of a microstrip line, where the parameters **w** and **h** are the width of microstrip line and the height of substrate for **w**<=**h**.

Calculate the characteristic impedance of a microstrip line, where the parameters **w** and **h** are the width of microstrip line and the height of substrate for **w**>=**h**.

Calculate the characteristic impedance of a stripline, where the parameters **w** and **b** are the width of stripline and the distance between two PEC plates for **w** < 3.5**b**.

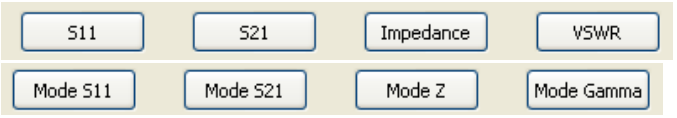
Calculate the characteristic impedance of a stripline, where the parameters **w** and **b** are the width of stripline and the distance between two PEC plates for **w** > 3.5**b**.

Calculate the cut-off frequency of a circular waveguide for the TE mode, where the parameter **a** is the radius of waveguide.

Calculate the cut-off frequency of a circular waveguide for the TM mode, where the parameter **a** is the radius of waveguide.

Calculate the cut-off frequency of a rectangular waveguide, where the parameters **a** and **b** are the width and height of waveguide.

GEMS pre-set hot buttons

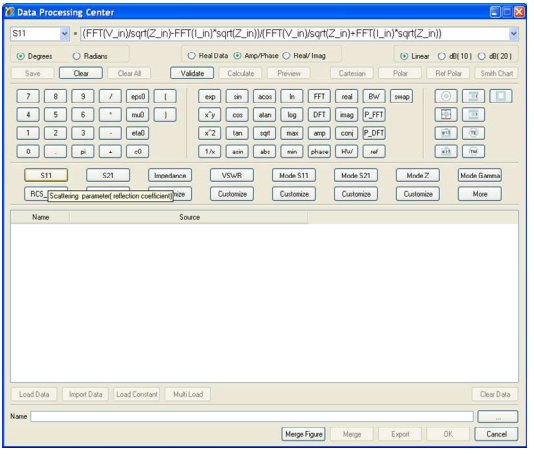


Click those buttons and the formulation is displayed in the formula box.

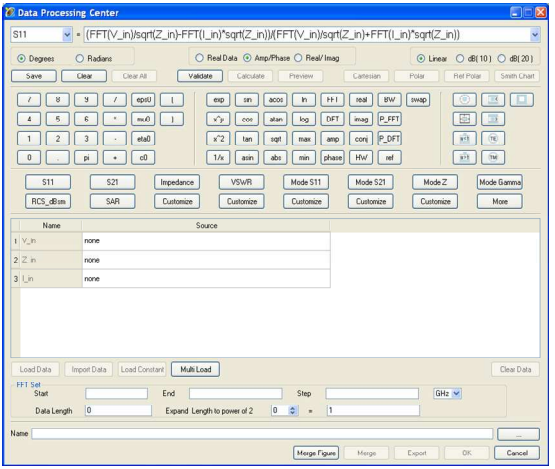
10.2 Basic Steps to Derive New Result

The basic steps to derive a result from the GEMS direct outputs are given below:

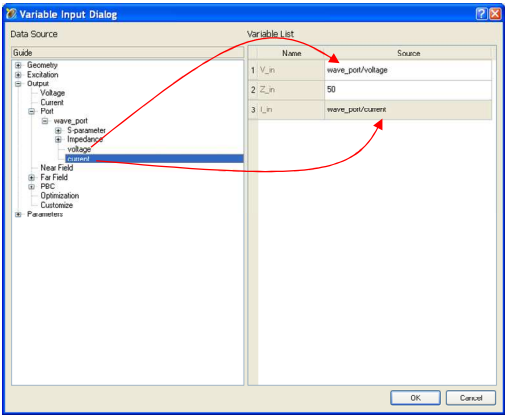
(1) Type a variable name and expression, or, click on one of the hot buttons, or, click on the **More** button to select one formula from the GEMS standard formula library. For example, click on the **S_{11}** button.



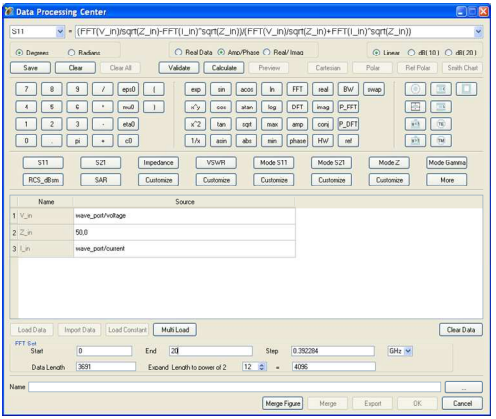
(2) Click on the **Validate** button to check the expression, and the validated variables will be listed in the variable table.



(3) Click on the **MultiLoad** button to assign the GEMS result to the variables

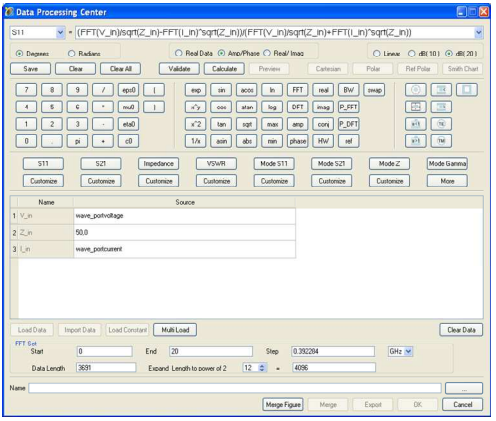


(4) Specify the frequency range of interest. You can pad more zeros by increasing the length of data to increase the frequency resolution.

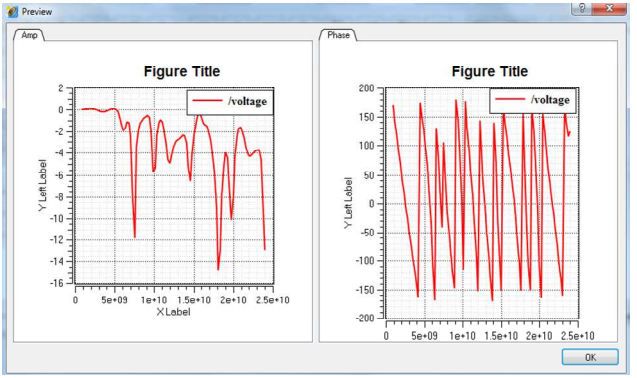


The variable S_{11} can be used as a new variable in other expressions to derive the new results. The available variables are listed in the variable list. You can clear the variable list.

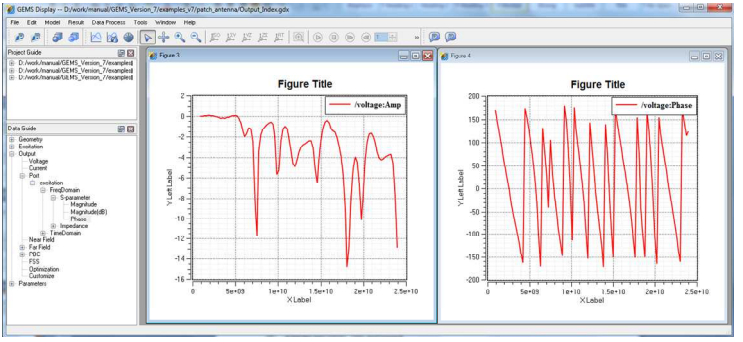
(5) Click on the **Calculate** button, select the display scale and result format.



(6) Click on the **PreView** button to preview the result



(7) Click on the **Cartesian** button to plot the result in the main picture region

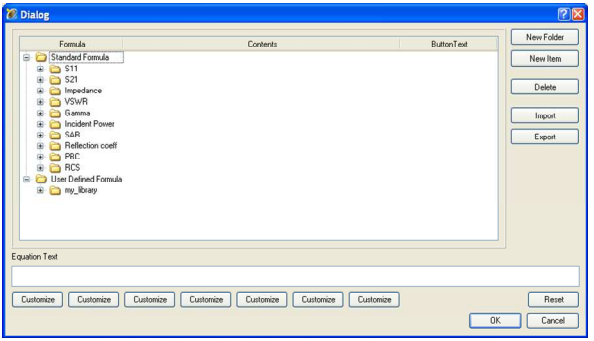


(8) Add the axis labels, title and legend.

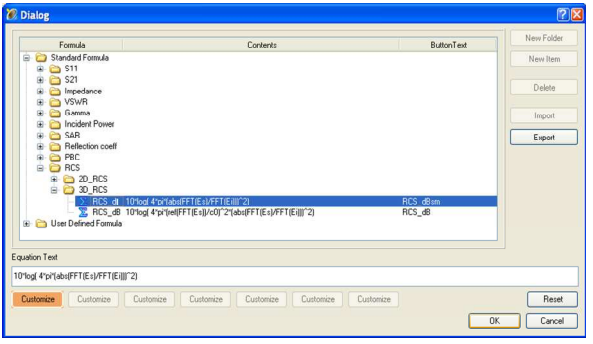
(9) Select the **Copy Figure** option in the **Edit** menu to copy the selected figure to the clipboard

10.3 Formula Library

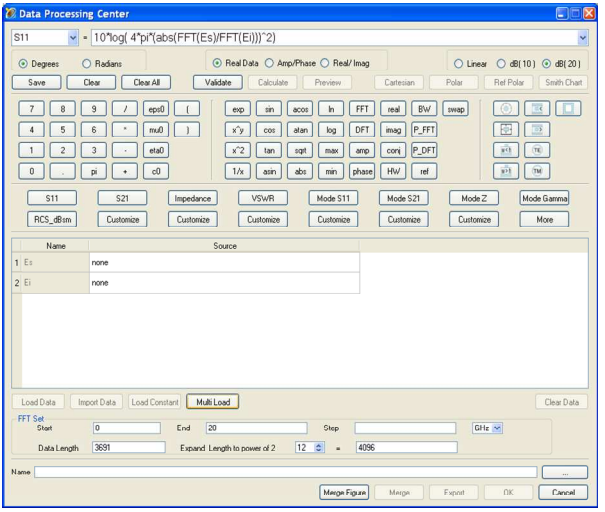
GEMS provides a standard formula library, and allows you to build their own formula library. You can also define the seven hot buttons. Click on the **More** button to open the formula library.



Select one of formulas from the library and click on it, and then click on **OK** button to load the formula to the formula box. You can also define it as a hot button by clicking on the **Customize** button. For example, select the RCS calculation formula from the standard library.



Click on the first **Customize** button, and then on the **OK** button.


Data Processing Center
S11
= 10*log(4*pi*(abs(FFT(Es)/FFT(Ei)))^2)
Degrees
Radians
Real Data
Amp/Phase
Real/ Imag
Linear
dB(10)
dB(20)
Save
Clear
Clear All
Validate
Calculate
Preview
Cartesian
Polar
Ref Polar
Smith Chart
S11
S21
Impedance
VSWR
Mode S11
Mode S21
Mode Z
Mode Gamma
RCS_dBsm
Customize
More
Name
Source
1 Es
none
2 Ei
none
Load Data
Import Data
Load Constant
Multi Load
Clear Data
FFT Set
Start
0
End
20
Step
GHz
Data Length
3691
Expand Length to power of 2
12
=
4096
Name
Merge Figure
Merge
Export
OK
Cancel

Index