Integrating CAD into the design process
Professor:

Jon D. Vredevoogd  vredevoo@msu.edu
TA's
Ashley Cowdrey  cowdreya@msu.edu
Colleen Crawford  crawf186@msu.edu

Office hours:

Directly following lecture; by appointment

Classrooms:

LECTURE:
MSU
108 Bessey
Tuesday  6:40-7:30pm

LAB:
Bessey Hall 216 has been reserved for several hours/week

Catalog Description:

Application of computer-aided design and structure principles in generating design solutions.

Course Objectives:

This course has been created for anyone who wants to understand and apply the design process to a small project. Emphasis is placed on the creativity of the solution. The solution will be developed with the assistance of CAD and it will introduce the student to structural design considerations and their evidence in the solution.

The student will design an exhibit to demonstrate his/her design knowledge and ability to execute the design process with the integration of CAD. The various tasks required to complete the project are clearly spelled out in the Table of Contents and in this document.

This course syllabus is a legal agreement between the faculty and student. We are each bound by the responsibilities stated in this document.
Materials:

**REQUIRED**

*Texts:*

  - The text includes:
    - Written Instruction
    - Drawing files on CD/Angel
    - Movies
    - Course scan sheets

*Course materials:*

- Headset for MSN
- CD

**RECOMMENDED**

*Text*

- *Using AutoCAD In Your Job: Part I*, (2006), Diane Bender & Jon D. Vredevoogd

*Course materials*

- Your personal copy of AutoCAD R2006 for your personal home computer
  - (Student Editions are available - see Jon)

Lab hours:

Teaching Assistants are available in CAD labs on campus 2 hours/M-F
Teaching Assistants are also available on-line 2 hours/M-F.
Lab schedules are posted on angel

Assignments

Assignments are issued so that you can get feedback on your work. You will:

a. Be able to see where your project stands among the submissions that week
b. Be able to learn from the class submissions how to improve your project
c. Hear comments from the faculty about the best submissions

Assignments are issued to make certain you complete an important task that must be finished before you can continue with the project
CLASS ORGANIZATION

Completed work

- ASSIGNMENT FOLDER
- PORTFOLIO FOLDER
- 309
  HE

- A FILE A FROM MULTIPULE FILES

- CRITIQUE FOLDER
  FILE FROM MULTIPULE FILES
  AUDIO
Transfering Files

File Transferring is the process of preparing a .pdf format image of your work and sending it to the appropriate ANGEL site.

We will be saving 3 types of files to the course site throughout the semester.

- **.pdf** These files are Adobe Portable Document files. They will make up almost all of our files. They save the work as an image.
  - To prepare a .pdf file you simply choose to print your file to a .pdf printer
  - Print the file in Landscape format - not Portrait

- **.dwg** These files are AutoCAD drawing files - You will send your drawing file only twice during the semester.
  - A drawing file of your final exhibit
  - A drawing file of your final cabinet.
  - If your .dwg file contains xref files and/or links - you will need to send all those files as well

- **.plt** These files are AutoCAD plot files - Two plot files will be created
  - A plot file of your final exhibit
  - A plot file of your final cabinet
  - When you plot your work to a plotter, a .plt file is created - that file is the one that is sent to the course site

The files are named using your last name followed by the file number (The task number)

For example:

- I would save my first file (TASK) as:
  - vredevoogd 01.pdf
- I would save my second file (TASK) as:
  - vredevoogd 02.pdf

**ASSIGNMENTS**

There is one task to be submitted each week

**PORTFOLIO**

This file folder contains a file for each student enrolled

You submit a second copy of your work in your PORTFOLIO folder:

- **309 HE**
  - You submit a scan sheet with your work printed on it (similar toIDES 240)
  - You will want to save one copy to your ZIP disk and/or a CD.
  - You can transfer your work any time before 4:00PM on the SUNDAY prior to class (Refer to the course outline)
  - You will find the ASSIGNMENTS and the PORTFOLIO file folders in the course angel site (Drop boxes)

**CRITIQUE**

All critiques and grading are based on the transferred files and the scan sheet
Attendance

Class time is a time to share information. It’s a time to interact with the faculty and TAs. It’s a time to talk about what worked and what didn’t. This is not a time to ask the faculty to critique work that was critiqued on the audio.

There will be in-class activities, which require the student’s participation to complete a phase of the project.

Late Work Policy

Late work is not accepted. Your work is evaluated in relation to the total of the class submissions making it easy to see where your project fits and impossible to evaluate individual projects after the due date.

If you cannot complete an assignment you must submit whatever you have completed at the due date.

Under extenuating circumstances, with proper written documentation, (i.e. illness = Physicians report, Death in the family = Obituary notice), the due date will be extended.

Grading

Your grade in the CLASS is based on the sum of the weekly TASKS submitted

Assignments are graded on using both an assignment checklist and the faculty’s emotional response to your project as it relates to the total number of projects submitted.

Each task is valued at 10 points - the assignments receive either full credit, 1/2 credit, or no credit.

Here is the final grade scale - using a possible 150 points for the class

<table>
<thead>
<tr>
<th>Points</th>
<th>Grade</th>
</tr>
</thead>
<tbody>
<tr>
<td>135</td>
<td>4.0</td>
</tr>
<tr>
<td>120</td>
<td>3.5</td>
</tr>
<tr>
<td>105</td>
<td>3.0</td>
</tr>
<tr>
<td>90</td>
<td>2.5</td>
</tr>
<tr>
<td>75</td>
<td>2.0</td>
</tr>
<tr>
<td>60</td>
<td>1.5</td>
</tr>
<tr>
<td>45</td>
<td>1.0</td>
</tr>
</tbody>
</table>

Cheating

This course adheres to the policies on academic honesty as specified in General Student Regulations 1.0, Protection of Scholarship and Grades, and in the all-University Policy on Integrity of Scholarship and Grades. Cheating is not tolerated.

Disability

Note: “Students with disabilities should contact OPHS (Disability Resource Center).
I’ve taught traditional studio courses for many years on this campus and I, along with other faculty, have observed some inherent problems.

Typically, the first project that the faculty critiques get the most complete critique - - by the time the faculty get to number ten, they find themselves doing a lot of repeating - - and critiques get sketchier and sketchier. The first critique gets the most time and the last ones are quickly reviewed to meet the end of class deadline.

Students may have to wait two or more hours to get their 5 minutes with the faculty.

Some students position themselves to spend a lot of time working one-on-one with the faculty, asking for opinions and ideas. Sometime it’s hard to tell if it’s the student’s project or the faculty’s project.

If there are several sections of a course and these sections have different instructors, we find that the success of the project is more often dependent on whose section you were in rather than the capability of the student.

Here’s another problem. Students don’t benefit from seeing other student’s work and hearing the evaluation of that work until the projects are complete.

Here’s another consideration. Faculty get frustrated trying to evaluate projects that are so sketchy that a critique is almost meaningless. In the early stages of a project, we may see only one or two good examples - - the wholeclass could benefit from seeing those examples and hearing the critique of them.

Design projects are usually evaluated based on what’s submitted at the end of the project. If you, as a student, do not see the work of your peers, how can you tell where you stand in the class? Only the faculty see the big picture.

I could go on . . .

You will be required to upload your work almost weekly during the semester.

During the scheduled lecture time, we will all get the opportunity to discuss all the submissions. No surprises. You won’t have to have someone tell you whether your project is one of the better projects or the one of the worst projects - - you can see for yourself.

Because the projects will be presented back to the class in an Acrobat PDF format, you can remain somewhat anonymous. Reviewing the projects in the presentation gives you the opportunity to get ideas from the entire class. An idea or suggestion by the faculty to one students becomes available to all. If you see or hear a great idea, use it.

Now the class becomes dynamic - - the best project one week may no longer be the best project the next. This is the way it is in the real world of professional design.

How exciting!
How to get a good grade in this class:

1. Understand the rules and play by the rules.

   *This class gives the highest scores to the most innovative, unique, and workable solutions* - unlike other classes that give the highest scores to the project that meets all the requirements. Here the emphasis of the class is on meeting the requirements of the check list followed by the “uniqueness factor”.

   Companies buy things because THEY LIKE THEM - - THEY WILL SET THE COMPANY APART FROM THEIR COMPETITION they count on the fact that details can be worked out later.

2. Stay on schedule.

   This class is divided into 15 tasks. Each task contains several subtasks. The tasks are presented in a specific order. Complete and check off each task until the project is complete.  *You can work ahead*

3. Always submit something

   You will find yourself getting stuck on a task - - complete what you can, submit what you have and move on.

   Use the critique to get direction

4. Plan on the fact that the project will take longer then you think it will take

   Give yourself time to go back and make changes - -

   NO DESIGNER SITS DOWN AND WORKS HIS WAY FROM THE BEGINNING TO THE END WITHOUT HAVING TO GO BACK AND MAKE CHANGES IN AN EARLIER PART OF THE PROJECT. HE/SHE THEN CHANGES EVERYTHING FROM THAT POINT FORWARD - - YOU WILL BE NO DIFFERENT.

5. Be working at least one presentation ahead of schedule

   Get your work done early - - nothing works perfectly - - there will always be unexpected challenges - - equipment failure, absent employees, illness - - learn to accept them as challenges - -

6. Take advantage of every opportunity to get feedback

   Most projects fail for lack of communication - not design ineptness

**GET YOUR WORK DONE EARLY & BE CREATIVE.**
<table>
<thead>
<tr>
<th>Task</th>
<th>Date</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Task 1</td>
<td>1/10/06</td>
<td>Communication skills</td>
</tr>
<tr>
<td>Task 2</td>
<td>1/17/06</td>
<td>Project overview &amp; exhibit selection</td>
</tr>
<tr>
<td>Task 3</td>
<td>1/24/06</td>
<td>Solid Surfaces</td>
</tr>
<tr>
<td></td>
<td></td>
<td>box, pyramid, wedge, dome, sphere, cone, toris, dish, mesh 3D face, 3D mesh</td>
</tr>
<tr>
<td></td>
<td></td>
<td>ai_dish, ai_dome, properties, ai pyramid, edge surf, pface, revsurf, rulesurf</td>
</tr>
<tr>
<td>Task 4</td>
<td>1/31/06</td>
<td>Solid Objects</td>
</tr>
<tr>
<td></td>
<td></td>
<td>3d_box, rotate3d, 3darray, cylinder, cone, sphere, Isolines, torus, facetres</td>
</tr>
<tr>
<td></td>
<td></td>
<td>wedge, mirror3d, extrude, revolve, slice, intersect, subtract, union,</td>
</tr>
<tr>
<td>Task 5</td>
<td>2/07/06</td>
<td>interference, tabulated, ruled, and dish, mesh, edge, revolve, section</td>
</tr>
<tr>
<td>Task 6</td>
<td>2/14/06</td>
<td>Exhibit in 3d solids</td>
</tr>
<tr>
<td>Task 7</td>
<td>2/21/06</td>
<td>Foundation, Floor, Wall</td>
</tr>
<tr>
<td>Task 8</td>
<td>2/28/06</td>
<td>Window, Door, Baseboard, Roof</td>
</tr>
<tr>
<td>Task 9</td>
<td>3/14/06</td>
<td>Directives &amp; Concepts</td>
</tr>
<tr>
<td>Task 10</td>
<td>3/21/06</td>
<td>Exhibit Preliminary</td>
</tr>
<tr>
<td>Task 11</td>
<td>3/28/06</td>
<td>Exhibit final</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Rendering</td>
</tr>
<tr>
<td>Task 12</td>
<td>4/04/06</td>
<td>Drawing a Cabinet</td>
</tr>
<tr>
<td>Task 13</td>
<td>4/11/06</td>
<td>Cabinet elevations &amp; sections</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Elevations and sections</td>
</tr>
<tr>
<td>Task 14</td>
<td>4/18/06</td>
<td>Cabinet Dimensions</td>
</tr>
<tr>
<td>Task 15</td>
<td>4/25/06</td>
<td>Cabinet final</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Hardware</td>
</tr>
</tbody>
</table>
Task [1]

Communication Tools
The task this week:
Learn the course communication tools
Submit Task 1 Communication scan sheet

To complete this task, you will need to:
Complete the communications tasks (Use the instruction movies)
Make on-line contact with a TA to check your screen
Print your completed Task 1 scan sheet to a pdf printer
Submit one copy to assignments, one copy to your portfolio and your scan to HE 309

The tools include
movies
TA support in labs and on-line
Task [1] scan sheet

Instructions follow
1. Uploading files to the course site
   - Assignments
   - Portfolio

2. Downloading from the course site
   - Critiques (pdf files)
   - Audio files (mp3 files)
   - Movie files (Camtasia self extracting files)

3. Installation of required software
   - MSNmessenger
   - Primopdf
   - SMARTftp

4. Operate required software
   - Access the Course Angel site
     - MSNmessenger
       - Access the MSNmessenger site
       - Add a TA contact
       - Make an audio connection
       - Share an application
       - Obtain remote assistance
       - Transfer files
     - PDF software printer
       - Save your work in a PDF format
     - CD burner
       - Save your work to a CD
Task [2]

Exhibit Project
The task this week:
Surf the internet and capture an image of an interesting exhibit
Something interesting from both a design and content perspective
Submit a Task 2 scan sheet (2 copies to angel sites - original to 309 HE)

To complete this task, you will need to:
- You need to find an exhibit - (check the example above)
- The two criteria that will be judged will be
  - its uniqueness
  - its use of the design language
  - the importance of its content
  - the depth of the idea
- Print your completed Task 1 scan sheet to a pdf printer
- Submit one copy to assignments, one copy to your portfolio and your scan to HE 309

The tools include
- movies
- TA support in labs and on-line
- Task blank scan sheets

Instructions follow
Task [3]

3D Solid **Surfaces**
The task this week is to learn to construct and use 3d surfaces

- ai_box
- ai_pyramid
- ai_wedge
- ai_dome
- ai_sphere
- ai_cone
- ai_torus
- ai_dish
- ai_mesh
- 3dmesh
- properties
- edgesurf
- pface
- revsurf
- rulesurf
- solid (2d)
- tabsurf
- 3dface

To complete this week’s task, you will need to:

Make certain you have set enough time aside to complete the work,
Save each construction as a block and then insert them into model space
Place your completed work in the correct locations

The tools include

- The course pack instructions
- Movies
- AutoCAD help menu
- TA support on line and in labs
3D SURFACES

The 3D command produces only shapes defined by their SURFACE only. In combination with commands that apply surfaces to existing wire frame drawings, you can create many complex shapes. First, draw the object in 2D using drawing and editing commands. (It is easier to make changes in 2D than in 3D). Then bring it into 3D with commands such as copy, move, 3Darray, etc. by adding the third coordinate, Z. Then apply a polygon mesh to the wire frame to form a solid surface. This is comparable to putting up a building’s framing system and then covering it with surfaces like plywood and gypsum board. By setting the two system variables SURFTAB1 and SURFTAB2 to different numbers, you can achieve a desirable level of smoothness. (The higher the number, the smoother the surface).

3d

Creates three-dimensional polygon meshes

The 3d command will provide a list of 9 solid surface objects. To access these from the keyboard, type ai_ before the object name.

- Box
- Cone
- Dish
- Dome
- Mesh
- Pyramid
- Sphere
- Torus
- Wedge

These are NOT solid objects, but 3d wire frame drawings with solid surfaces attached to their planes. For example, the BOX command will create a solid box that you can manipulate with commands like slice, section, union, etc. But the 3DBOX command will produce a box with the appearance of solidity, but cannot be manipulated for solid modeling.

ai_box

Creates a three dimensional box polygon mesh

- This 3d box is not to be confused with a solid box
- This creates a three-dimensional polygon mesh box

3d <-
[Box/Cone/Dish/Dome/Mesh/Pyramid/Sphere/Torus/Wedge]:
Specify the corner point of the box 0,0,0
Specify the length of the box 2"
Specify the width of the box [or cube] 1"
Specify the height of the box .5"

If your are prompted for the rotation around the ’Z’ axis, specify 0
**Ai_pyramid**  
*A pyramid or tetrahedron polygon mesh*

- This 3d object can have a square (pyramid) or triangular base (tetrahedron).
- It may be easiest to define the base points and top point(s) with absolute coordinates.
- The shape of the top has many options and all must have a Z value to determine the height of the object.
- Height options:
  - **Ridge**: Defines the pyramid top as a ridge line by specifying first and second ridge points. These points should be in the same direction as the pyramid’s base points.
  - **Top**: Defines the pyramid or tetrahedron top with either three points or four points (truncated). Make sure the top points do not cross each other or you will have a self-intersecting wire frame.
  - **Apex point**: Default option that defines the pyramid or tetrahedron top.

```
3d <-
[Box/Cone/Dish/Dome/Mesh/Pyramid/Sphere/Torus/Wedge]: p-
Specify first corner point for base of pyramid: specify 0,0,0
Specify second corner point for base of pyramid: specify 2,0,0
Specify third corner point for base of pyramid: specify 2,2,0
Specify fourth corner point for base of pyramid or [Tetrahedron]: specify a point complete the pyramid or choose T for tetrahedron 0,2,0
Specify apex point of pyramid or [Ridge/Top]: specify 1,1,2
```

*Note that the apex needed to be specified with 3 co-ordinates to locate the apex in the center of the pyramid*

**ai_wedge**  
*A wedge polygon mesh*

- Creates a right-angle, wedge-shaped polygon mesh with a sloped face tapering along the X axis.

```
Specify corner point of wedge (point 1) 0,0,0
Specify length of wedge 2
Specify width of wedge 1
Specify height of wedge .5
Specify rotation angle of wedge about the Z axis 0
```
The dome's size can be defined by either its radius or its diameter.
Longitudinal segments are radial; latitudinal segments are along the Z axis.
The higher the number of segments, the more fluid the dome. This increases the amount of regeneration time for the object.

3d <-
[Box/Cone/Dish/DOme/Mesh/Pyramid/Sphere/Torus/Wedge]: DO<-
Specify center point of dome: 0,0,0
Specify radius of dome or [Diameter]: d
Accept the following defaults
Enter number of longitudinal segments for surface of dome <16>: enter a value greater than one
Enter number of latitudinal segments for surface of dome <8>: enter a value greater than one

I find this command a little touchy - sometimes if reads a radius as a diameter - specifying what you want as we did above often corrects it
The radius should be the default

The sphere's size can be defined by either its radius or its diameter.
Longitudinal segments are radial; latitudinal segments are along the Z axis.
The higher the number of segments, the more fluid the dome. This increases the amount of regeneration time for the object.

3d <-
[Box/Cone/Dish/DOme/Mesh/Pyramid/Sphere/Torus/Wedge]: s<-
Specify center point of dome: 0,0,0
Specify radius of dome or [Diameter]: 1
Enter number of longitudinal segments for surface of dome <16>: enter a value greater than one
Enter number of latitudinal segments for surface of dome <8>: enter a value greater than one

The cone's size can be defined by either its radius or its diameter.
Radius base defines the base of the cone by its radius
Radius top defines the top of the cone by its radius

3d <-
[Box/Cone/Dish/DOme/Mesh/Pyramid/Sphere/Torus/Wedge]: DO<-
Specify center point of dome: specify 0,0,0
Specify radius for the base of the cone or [Diameter]: 1
Specify radius for the top of the cone or [Diameter]:<0> 0
Enter number of longitudinal segments for surface of dome <16>: enter a value greater than one
Enter number of latitudinal segments for surface of dome <8>: enter a value greater than one
**Ai_torus**  
*Creates a toroidal polygon mesh that is parallel to the XY plane of the current UCS*

- Check the AutoCAD help menu to really understand the terms.

```
3d <-
Specify center point of torus: specify a point 0,0,0
Specify radius of torus or [Diameter]: enter a value .75
  The radius of the torus is measured from its center point to its outer edge - not to the center of the tube
Specify radius of tube or [Diameter]: enter a value .25
Enter number of longitudinal segments for surface of dome <16>: enter a value greater than one
Enter number of latitudinal segments for surface of dome <8>: enter a value greater than one
```

**Ai_dish**  
*The lower half of a spherical polygon mesh*

- The dish’s size can be defined by either its radius or its diameter.
- Longitudinal segments are radial; latitudinal segments are along the Z axis.
- The higher the number of segments, the more fluid the dish. This increases the amount of regeneration time for the object.

```
3d <-
[Box/Cone/Dish/Dome/Mesh/Pyramid/Sphere/Torus/Wedge]: DI <-
Specify center point of dish: 0,0,1
Specify radius of dish or [Diameter]: 1
Enter number of longitudinal segments for surface of dish <16>: enter a value greater than one
Enter number of latitudinal segments for surface of dish <8>: enter a value greater than one
```

**ai_mesh**  
*A planar mesh*

- Creates a coplanar mesh whose M and N sizes determine the number of lines drawn in each direction along the mesh.
- The M and N directions are similar to the X and Y axis of an X Y plane
- Note that the 4 points form a rectangle even when you twist the mesh

```
Specify first corner point of mesh: Specify a point (1) 0,0,0
Specify second corner point of mesh: Specify a point (2) 0,15/16,9/16
Specify third corner point of mesh: Specify a point (3) 2,0,0
Specify forth corner point of mesh: Specify a point (4) 2,0,2
Enter mesh size in the M direction: Enter a value between 2 and 356 30
Enter mesh size in the M direction: Enter a value between 2 and 356 10
```
**3dmesh**

A free-form polygon mesh

- 3Dmesh creates a complex polygon mesh defined by a matrix.
- Its size is determined by the number of vertices that you specify. M (the rows) and N (the columns) are multiplied together \((M \times N)\) to get the number of vertices.
- The distance between vertex points is not necessarily constant; it is determined by the vertex points you enter.
- It is easiest to enter absolute coordinates for vertex points.
- The higher the value entered for both M and N, the more vertex points will be required.

**3dmesh <-**

Enter size of mesh in M direction: *enter a value between 2 and 256* 5
Enter size of mesh in N direction: *enter a value between 2 and 256* 5
Specify location of vertex \((m,n)\): *specify a 2D or 3D coordinate*

AutoCAD will continue to prompt you for vertex points equal to the product of the numbers you entered for the M and N sizes \((M \times N)\). The vertex points are entered as absolute co-ordinates (here are the co-ordinates you will need):

<table>
<thead>
<tr>
<th>M direction</th>
<th>4,0,0</th>
<th>4,1,0</th>
<th>4,2,0</th>
<th>4,3,0</th>
<th>4,4,0</th>
</tr>
</thead>
<tbody>
<tr>
<td>N direction</td>
<td>3,0,0</td>
<td>3,1,0</td>
<td>3,2,0</td>
<td>3,3,1</td>
<td>3,4,0</td>
</tr>
<tr>
<td></td>
<td>2,0,0</td>
<td>2,1,0</td>
<td>2,2,1</td>
<td>2,3,0</td>
<td>2,4,0</td>
</tr>
<tr>
<td></td>
<td>1,0,0</td>
<td>1,1,1</td>
<td>1,2,0</td>
<td>1,3,0</td>
<td>1,4,0</td>
</tr>
<tr>
<td></td>
<td>0,0,0</td>
<td>0,1,0</td>
<td>0,2,0</td>
<td>0,3,0</td>
<td>0,4,0</td>
</tr>
</tbody>
</table>

**Properties**

Changes the properties of an existing object

- These commands alter an entity’s location, identity and appearance.
- The parameter changes depend on the type of entity you pick (i.e. lines, circles, text, attribute definitions, or blocks).
- Make sure you know where the UCS is located before using the thickness option.
- Five options are available:
  - **Color** Alters the color of an object regardless of its layer
  - **Layer** Moves an object from one layer to another
  - **Linetype** Alters an object’s linetype appearance
  - **Linetype Scale** Alters an object’s linetype scaling; independent of the LTSCALE command
  - **Thickness** Pulls an object into a solid plane along the Z axis

Specify center point for circle or \([3P/2P/Ttr (tan tan radius)]: 0,0,0\)

Specify radius of circle or \([\text{Diameter}] <0'-0\ 1/4'>: 1\)

**Properties <-**

Select the circle,

The properties dialog box will appear. Select thickness and change 0" to 2"
Edgesurf

A three-dimensional polygon mesh

- Edgesurf is similar to a mesh.
- Edgesurf uses four adjoining edges to create a polygon mesh that approximates a surface patch.
- This bicubic surface meets and touches the corners of the defining edges, which allows for greater control over the boundaries of the generated surface patch.
- The four edges may be lines, arcs, or open 2d or 3d polylines and can be selected in any order.
- Be sure that the edges touch at their endpoints to form a rectangular closed path.
- The first edge selected determines the M direction of the mesh while the two edges that run perpendicular to the first edge define the N direction.
- The commands SURFTAB1 and SURFTAB2 regulate the smoothness of the mesh and the distance between the mesh vertices for the M and N directions.

Edgesurf <-
Current wire frame density:  SURFTAB1=12    SURFTAB2=6
Set wire frame using the commands
surftab1<-
surftab2<-
Select object one for surface edge: select an object
Select object two for surface edge: select an object (the sequence of selection is not important)
Select object three for surface edge: select an object
Select object four for surface edge: select an object

Pface

A three dimensional polyface mesh created vertex by vertex

- Pface allows you to create an unusually shaped mesh that can have visible or invisible edges.
- You specify each vertex and then associate it with faces in the mesh. You first specify all vertices in the mesh by specifying a point or pressing return. (You must know these points before beginning). When you press return, AutoCAD prompts you to assign the vertex to a face.
- To make an edge invisible, enter a negative vertex number for the beginning vertex of the edge.
- Define each face by entering the vertex numbers that define that face. When you are ready to proceed to the next face, press enter and you will be prompted for the vertex numbers of the next face. Press enter on a blank line after you have defined the last face.

Pface <-
Specify location for vertex 1: specify a point
Specify location for vertex 2 or<define faces>: specify a point or press enter
Specify location for vertex n: specify a point or press enter
Face 1, vertex 1:
Enter a vertex number or [Color/Layer]: enter color, layer, or a vertex number
Face n, vertex n:
Enter a vertex number or [Color/Layer]: enter color, layer, a vertex number or enter
Revsurf

Creates a rotated surface about a selected axis

- For revsurf, you will need two things: a path curve and an axis to rotate around.
- It is easiest to make the object you would like to rotate as an enclosed polyline.
- You can change a combination of lines and arcs into a continuous polyline by using the PEDIT command.
- You will most likely rotate around the Z axis, therefore, take note of your current UCS.
- It may be helpful to do this command by looking at more than one viewport (VPORT).
- The distance between the path curve (object to rotate) and the object that defines the axis of rotation will dictate how far the revsurf occurs from the axis line.
- A positive angle of revolution makes the object go counter-clockwise. A negative angle of revolution makes the object go clockwise.

Revsurf <-
Select object to revolve: select the object you would like to have rotated in 3D
Select object that defines the axis of revolution: select the line that you will rotate around (this is probably in the Z direction)
Specify start angle (+ccw, -cw) <360>: type a new angle or accept 0 as the default
Specify included angle <full circle>: type an angle or accept full circle as the default

Rulesurf

Creates a ruled surface between two curves

- Rulesurf creates a polygon mesh in the form of a ruled surface between two boundaries of nearly any type.
- The two different entities that define the boundaries can be points, lines, arc, circles and 2D or 3D polylines.
- If one boundary is closed, then the other must also be closed.
- Selecting objects at the same ends produces a polygon mesh, while selecting opposite ends produces a self-intersecting polygon mesh.
- The number of intervals is determined by the command SURFTAB1. This regulates the distance between the mesh vertices.

Rulesurf <-
Current wire frame density SURFTAB1=#
Select first defining curve: select the first object
Select second defining curve: select the other object
**Solid**

*Creates solid-filled polygons*

- A solid is similar to a 3D face except for its filled surface and its edge defining sequence.
- To fill a solid, the FILLMODE system variable must first be set to ON.
- If a type of quadrilateral area is desired, the four points must be selected out of order, crisscrossing like a bow tie. An incorrect selection sequence is a common mistake for first-time users.

**Solid**

- Specify first point: *specify a point (use OSNAP options)*
- Specify second point: *specify a point*
- Specify third point: *specify a point diagonally opposite the second point*
- Specify fourth point or <exit>: *press enter for a triangular solid; specify a point for a quadrilateral solid*

AutoCAD will repeat the third point and fourth point prompts until you press enter. By specifying further third and fourth points, connected triangles and polygons are added to this single solid object.

**Tabsurf**

*Draw Surfaces 2D Solid*

*Draw Surfaces Tabulated surface*

- To create a tabulated surface, you need to first define a path curve and a direction vector. The path curve defines the surface for the polygon mesh. The direction vector is a line or pline that indicates the length of the extruded polygon mesh.
- Where you select on the direction vector will tell AutoCAD which direction it will be extruded.
- For a 3-dimensional result, have the path curve parallel to the current UCS and have the direction vector projecting along the Z axis.
- The direction vector may be at an angle or have many vertices in between the first point and endpoint, though AutoCAD will ignore these intermediate vertices when extruding. If you are using a polyline with many intermediate vertices, you may choose to spline it with the PEDIT command to get a more fluid and flowing polyline. Otherwise, AutoCAD will draw tabulated lines at the ends of each straight segment.

**Tabsurf**

- Select object for path curve: *select object*
- Select object for direction vector: *select object*
- Complete the instructions in the following sections of
3dface

A three-dimensional face selected at four points

- 3Dface creates a 2-dimensional solid surface.
- It is easiest to begin with a wire frame drawing and then apply a face using your OSNAP options to pick the corners.
- Unlike solid, you can specify a point with any Z coordinate. (It does not have to be parallel to the current UCS).
- The variable Points must be selected in a clockwise or counterclockwise order and not across diagonally.
- When hidden, shaded or rendered, these planar faces are filled.

**3dface**<-

Specify first point or [Invisible]: *specify a point* (use OSNAP options)
Specify second point or [Invisible]: *specify a point*
Specify third point or [Invisible]<exit>: *specify a point* (do not cross over diagonally like a SOLID)
Specify fourth point or [Invisible]<create three-sided face>: *specify a point*
( Specify third point or [Invisible]<exit>: <-

AutoCAD will continue prompting you for a third point, a fourth point, a third point, etc. until you press Enter)
Task [4]

3D Solid **Objects**
The task this week is to learn to construct and use 3d surfaces

_box shelves table
_cone counter _sphere
_isolines _torus facetres
_wedge mirror3d _extrude
_revolve glass lamp

To complete this week’s task, you will need to:

Make certain you have set enough time aside to complete the work,
Place your completed work in the correct locations

The tools include

The course pack instructions
Movies
AutoCAD help menu
TA support on line and in labs
• Drawings usually progress through various stages such as wireframe drawing and then the application of solid surfaces. But the ultimate 3-dimensional experience is with 3d solid objects.
• 3d solids have mass and volume. They can be combined, intersected, and overlapped. In a later module, you will have the computer calculate sections with automatic hidden line placement and hatching.
• When you explode a solid, the sides of it return to regions or faces. If you explode one more time, the regions will turn into individual entities like lines and arc. When this happens, the object loses its 3dimensional solid properties. The best way to recover is to use the ‘undo’ command.
• 3D solids and solid surfaces can be rendered. You will render the objects you are creating in a later module.

CREATING SOLID BOXES
Open the file: box.dwg
Saveas: 3d_box.dwg

Box

A three-dimensional solid box

• A 3Dbox is a solid, six-sided form.
• The first corner of the box is the base point for rotation.
• A cube can be created by choosing C for cube and entering a value for the length.
• Once a box is created, you cannot alter its size by the stretch or change commands (use the scale command).
• The center option will allow you to locate a center point for the box. This is the center in the X, Y, and Z directions.
• When creating a rectangular box, the length is in the X direction, the width is in the Y direction and the height is in the Z direction. Be careful where your UCS is located when you begin!

SOLID CABINET FRAME
Constructing the shelving unit for the Tea House

Current layer: a-cab-frame

Create the cabinet base without the trimwork. You will construct it standing on its end and then rotating it in 3D space.

Box
Specify corner of box or [Center]<0,0,0>: 44'4-13/16,2'6-3/4 <-
Specify corner of [Cube/Length]: L <-
Specify Length: 1'3 <-
Specify Width: 1'7 <-
Specify Height: 6'11 <-

**Rotate3d**

Rotates an object or series of objects in three-dimensional space

- A three-dimensional axis will need to be defined as your rotation axis. This can be defined by selecting points visually or with OSNAP options, or entering absolute coordinates.
- All options will request you to define a rotation angle or a reference angle.
- 4 options:
  - Axis by object: Aligns the axis of rotation with an existing object
  - Last: Uses the axis of rotation last selected
  - View: Aligns the axis of rotation with the current viewport’s viewing direction
  - 2points: Defines the axis of rotation with 2 specified points
  - XY/YZ/XZ: Aligns the axis of rotation with the X, Y, or Z axis, which passes through a selected point

---

**Rotate3d**

(Using the 2points option)

Select objects: I <- (type L for last)
Select objects: <-
[Object/Last/View/Xaxis/Yaxis/Zaxis/2points]: 2p <-
Specify first point on axis: with the endp OSNAP, pick point #1
Specify second point on axis: with the endp OSNAP, pick point #2
Specify rotation angle or [Reference]: -90 <- (don’t forget the negative sign!)

---

**Create the backing material for the cabinet.**

**Box**

Specify corner of box or [Center]<0,0,0>: 37'5-13/16, 2'6-3/4, 1'3 <-
Specify corner of [Cube/Length]: L <-
Specify Length: 6'11 <-
Specify Width: 3/4 <-
Specify Height: 6'6 <-

**Create the top for the cabinet.**

**Box**

Specify corner of box or [Center]<0,0,0>: 37'5-13/16, 2'6-3/4, 7'9 <-
Specify corner of [Cube/Length]: L <-
Specify Length: 6'11 <-
Specify Width: 1'7 <-
Specify Height: 3 <-

Hide <-
Regen <-
SOLID CABINET SUPPORTS

Create the one cabinet vertical support
Current layer: a-cab-supports

Box
- Specify corner of box or [Center]<0,0,0>: 37'5-13/16,2'7-1/2, 1'3
- Specify corner of [Cube/Length]: L
- Specify Length: 5-3/4
- Specify Width: 1'6-1/4
- Specify Height: 6'6

3darray
Create a three-dimensional array
- Rectangular: A rectangular array produces a number of rows, columns and levels in a matrix format.
- You must have at least one row, column, or level for this command to function properly (ex: bookcase shelving).

Create the remaining 3 vertical supports.
3darray
- Select objects: l (type L for last)
- Enter the type of array [Rectangular/Polar]<R>: 
- Enter the number of rows (---) <1>: 
- Enter the number of columns (|||) <1>: 4
- Enter the number of levels (…) <1>: 
- Enter the distance between columns (|||): 2'1-3/4

Hide
Regen

SOLID CABINET SHELVES

Create the shelves
Current layer: a-cab-shelves

Box
- Specify corner of box or [Center]<0,0,0>: 37'11-9/16,2'7-1/2,1'3
- Specify corner of [Cube/Length]: L
- Specify Length: 1'8
- Specify Width: 1'5-1/4
- Specify Height: 3/4

Issue a 3darray to duplicate the shelves.
This rectangular array will have 1 row, 3 columns and 5 levels.
Distance between columns: 2'1-3/4
Distance between levels: 1'3-3/4

Hide
Regen
Check the solution layer to determine your accuracy.
Qsave
**A SOLID TABLE**

Construct a table using 3D solid commands

*Zoom, extents in all viewports* <-

**Current layer:** table

**Freeze layer:** solution

Create the base of the table in two parts.

*Box* <-

- Specify corner of box or [Center]<0,0,0>: 46'7-7/8,4'10-11/16 <-
- Specify corner of [Cube/Length]: L <-
- Specify Length: 9 <-
- Specify Width: 9 <-
- Specify Height: 1-1/2 <-

Create the table support.

*Box* <-

- Specify corner of box or [Center]<0,0,0>: 46'10-7/8,5'1-11/16,1-1/2 <-
- Specify corner of [Cube/Length]: L <-
- Specify Length: 3 <-
- Specify Width: 3 <-
- Specify Height: 2'4-1/2 <-

Create the table top.

*Box* <-

- Specify corner of box or [Center]<0,0,0>: 46'1-7/8,4'4-11/16,2'6 <-
- Specify corner of [Cube/Length]: L <-
- Specify Length: 1'9 <-
- Specify Width: 1'9 <-
- Specify Height: 2-1/2" <-

Curve the edges of the table top.

*Fillet* <-

- Current settings: Mode=TRIM, Radius=0'0-1/2"
  - Select first object or [Polyline/Radius/Trim]: *pick one of the top edges* (#1)
  - Enter fillet radius <0'0-1/2": <-
  - Select an edge or [Chain/Radius]: *pick the edge below* (#2)
  - Select an edge or [Chain/Radius]: <-

*Continue with the fillet command to curve all 4 sides of your table.*

*Hide* <-

*Regen* <-

Check the solution layer to determine your accuracy.

*Qsave* <
CREATING SOLID CYLINDERS

Creating different parts of the circular countertop.
Complete the instructions in the following for constructing
the counter and overhead structure of the tea house

Open the file: cylinder.dwg
Saveas: 3d_cylinder.dwg
Current layer: cylinder

Cylinder

* Creates a solid 3D cylinder
* The elliptical option will form a cylinder with an elliptical base.
* This base can be defined by:
  3 points forming the major and minor axes
  A center point and the radius of each axis

These cylinders will overlap the wall. You will slice them later.

Cylinder

Specify center point for base of cylinder or [Elliptical] <0,0,0>: 32'1-1/16,24'3-9/16,3'7-1/2 <-
Specify radius for base of cylinder or [Diameter]: 14'6 <-
Specify height of cylinder or [Center of other end]: 1' <-

Cylinder

Specify center point for base of cylinder or [Elliptical] <0,0,0>: 32'1-1/16,24'3-9/16,4'7-1/2 <-
Specify radius for base of cylinder or [Diameter]: 14'8 <-
Specify height of cylinder or [Center of other end]: 1-1/2 <-

Cylinder

Specify center point for base of cylinder or [Elliptical] <0,0,0>: 32'1-1/16,24'3-9/16 <-
Specify radius for base of cylinder or [Diameter]: 12'3-3/8 <-
Specify height of cylinder or [Center of other end]: 4'9 <-

Hide
Regen
Check the solution layer to determine your accuracy.
Qsave
CREATING SOLID CONES

Creating the ceiling structure above the counter.

Open the file: cone.dwg
Save as: 3d_cone.dwg
Current layer: cone

**Cone**

A cone-shaped solid

- The cone's base can be defined by either its radius or its diameter.
- The elliptical option will form a cylinder with an elliptical base. This base can be defined by:
  - 3 points forming the major and minor axes
  - A center point and the radius of each axis
- 2 options for creating the top of a cone:
  - **Apex**: defines the height and orientation based on the current Z axis; the apex point may or may not be aligned with the center of the cone, thus allowing you to form skewed cones
  - **Height**: defines the height along the current Z axis; positive numbers go along the positive Z axis and negative numbers go along the negative Z axis (produces an upside-down cone)

Creating the structure above the counter requires a cylinder with a positive height and a cone with a negative height. The cone will point down into the counter. You will slice it off later.

**Cylinder**

Specify center point for base of cylinder or [Elliptical] <0,0,0>: 32'1-1/16,24'3-9/16,9'8-1/2 <-
Specify radius for base of cylinder or [Diameter]: 15'10 <-
Specify height of cylinder or [Center of other end]: 10 <-

**Cone**

Specify center point for base of cone or [Elliptical] <0,0,0>: with the center OSNAP, pick on the cylinder edge
Specify radius for base of cone or [Diameter]: 15'10 <-
Specify height of cone or [Apex]: -8' <- (don't forget the negative sign!)

Thaw layer: counter

Create a cup on the upper counter. Use a negative height to form the taper on the cup. You will slice the bottom off and subtract out the inside space later.

**Cone**

Specify center point for base of cone or [Elliptical] <0,0,0>: 39',12'6,5'5 <-
Specify radius for base of cone or [Diameter]: 2 <-
Specify height of cone or [Apex]: -8' <-

**Cone**

Specify center point for base of cone or [Elliptical] <0,0,0>: 39',12'6,5'5 <-
Specify radius for base of cone or [Diameter]: 1.95 <-
Specify height of cone or [Apex]: -7'6' <-

Hide <-
Regen <-
Thaw the solution layer to determine your accuracy and then freeze it again before uploading the file - See the example Qsave <-
CREATING SOLID SPHERES

Open the file: sphere.dwg
Save as: 3d_sphere.dwg
Current layer: sphere

**Sphere**  
*A spherical solid*
- The sphere can be defined by its center point and either its radius or its diameter.
- The sphere is created so its central axis is parallel to the Z axis of the current UCS.
- The center of the sphere is its location in the X, Y, and Z directions.

Since there is not an apparent example of a sphere in this drawing example, create a ball on the upper countertop.
*Sphere* <-
  Specify center of sphere <0,0,0>: 40',13'6,4'10-1/2 <-
  Specify radius of sphere or [Diameter]: 1.5 <-

Create yet another ball, with radius of your choice.
*Sphere* <-
  Specify center of sphere <0,0,0>: 41',12',4'10 <-
  Specify radius of sphere or [Diameter]: pick a number of your choice <-

**Isolines**  
*A system variable that determines the number of lines visible on a solid object*
- Isolines can be set from 0 to 2047.
- The default setting on most drawing files is 4.
- The higher the number, the smoother the appearance of your solid objects in hide, shade, or render. However, the smoother it looks, the longer the software will take to regenerate the drawing on screen.

Change the isolines to a higher number.
*Isolines* <-
  Enter new value for ISOLINES <#>: 20 <-

Hide <-
Regen <-
Check the solution layer to determine your accuracy.
Qsave <-
CREATING A SOLID TORUS

Open the file: torus.dwg
Save as: 3d_torus.dwg
Current layer: torus

**Torus**

*A donut-shaped solid*

- A torus is constructed parallel to the XY plane of the current UCS.
- The torus diameter or radius is measured from the center point to the outside edge. The tube diameter or radius is the distance between the inside and outside edges.
- The tube diameter or radius may be less than or equal to the torus diameter or radius. It may also be greater than the torus diameter, which is referred to as a self-intersecting torus.
- If both radii are positive and the tube diameter is greater than the torus diameter, the torus resembles a sphere with indentations at each pole.
- If the torus radius is negative and the tube radius is greater than the torus radius, the torus resembles a sphere pointed at each pole.

Create a clockframe on the wall behind the counter.

OSNAP: endpoint

**UCS**

Enter an option [New/Move/orthoGraphic/Prev/Restore/Save/Del/Apply/?/World]<World>: n <-
Specify origin of new UCS or [ZAxis/3point/0bject/Face/View/X/Y/Z/]<0,0,0>: 3p <-
Specify new origin point <0,0,0>: pick the lower left corner of the back wall (#1)
Specify point on positive portion of X-axis <#>: pick the X axis point (#2)
Specify point on positive-Y portion of the UCS XY plane <#>: pick the Y axis point (#3)

OSNAP: cancel endpoint

**Torus**

Specify center of torus <0,0,0>: pick any point on the wall with your mouse <-
Specify radius of torus or [Diameter]: 12 <-
Specify radius of tube or [Diameter]: 1/2 <-

UCS, world <-
Thaw layer: counter

Create a doughnut on the upper counter.

**Torus**

Specify center of torus <0,0,0>: 39’6,12’10,4’10 <-
Specify radius of torus or [Diameter]: 2 <-
Specify radius of tube or [Diameter]: 1 <-

Zoom closer to the little torus in any viewport.
Facetres  A system variable that controls the smoothness of solid objects and their hidden lines
- Facetres can be set from 0.01 to 10.
- The default setting on most drawing files is 0.5.
- The higher the number, the smoother the appearance of your solid objects in hide, shade or render.
  However, the smoother it looks, the longer the software will take to regenerate the drawing on screen.

Change the facetres to a higher number.
Facetres <-
  Enter new value for FACETRES <0.5000>: 2<-

Hide <-
Regen <-
Check the solution layer to determine your accuracy.
Qsave <-

CREATING SOLID WEDGES
Creating the supports under the customer counter top

Open the file: wedge.dwg
Saveas: 3d_wedge.dwg
Current layer: wedge

Wedge  A right angle wedge-shaped solid
- The sloped face tapers along the X axis.
- The first corner of the wedge is the base point for rotation.
- The base of a wedge is parallel to the XY plane of the current UCS.
- The height of a wedge can be in the positive or negative Z direction.
- The center option will allow you to locate a center point for the wedge. This is the center in the X, Y, and Z directions.
- The cube option will create a wedge with equal length sides.
- The sloping edge faces the first corner point that selected and tapers along the X axis.

Create the triangular support under the customer’s countertop.
Wedge <-
  Specify first corner of wedge or [Center] <0,0,0>: 17'2-3/4,24'-11/16,3'6 <-
  Specify corner or [Cube/Length]: L <-
  Specify length: -8 <-
  Specify width: -1 <-
  Specify height: 2'8 <-
Mirror3d  
* Mirrors objects around a plane
- The original object can be copied or deleted after the mirroring process is completed.
- If using the 3points option, it may be helpful to put the drawing in an isometric view.
- 6 ways to mirror:
  
<table>
<thead>
<tr>
<th>Method</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>3points</td>
<td>Define 3 points to form the mirroring plane</td>
</tr>
<tr>
<td>Object</td>
<td>Select an existing circle, arc or 2D polyline segment for the mirroring plane</td>
</tr>
<tr>
<td>Last</td>
<td>Continues mirroring by using the last selected mirroring plane</td>
</tr>
<tr>
<td>Zaxis</td>
<td>Select a point on a plane and a Zaxis point</td>
</tr>
<tr>
<td>View</td>
<td>Aligns the mirroring plane with the current view</td>
</tr>
<tr>
<td>XY/YZ/ZX</td>
<td>Aligns the mirroring plane by specifying a point in the X, Y, or Z directions</td>
</tr>
</tbody>
</table>

Mirror3d <-
(using the 3point option)
Select objects: *pick the wedge*
Select objects: `<-
Specify first point of mirror plane (3 points) or
[Object/Last/Zaxis/View/XY/YZ/ZX/3points] <3points>: `<-
Specify first point on mirror plane: *with the endp OSNAP, pick point #1* (transparent zoom is necessary)
Specify second point on mirror plane: *with the endp OSNAP, pick point #2*
Specify third point on mirror plane: *with the endp OSNAP, pick point #3*
Delete source objects? [Yes/No] <N>: *type Y and enter to delete the original wedge*

Array <-
Select objects: *pick the wedge*
Select objects: `<-
Rectangular or Polar array (R/P): p `<-
Base/<Specify center of array>: 32’1-1/16, 24’3-9/16 `<-
Number of items: 8 `<-
Angle to fill <360>: 132 `<-
Rotate objects as they are copied <Y>: `<-
accept the default for yes

Zoom, extents
Thaw layer: *i-door*

Create a small doorstop for the door by the counter.
Wedge <-
Specify first corner of wedge or [Center] <0,0,0>: *with the endpoint OSNAP, pick the end of the door* `<-
Specify corner or [Cube/Length]: L `<-
Specify length: 5 `<-
Specify width: 1.5 `<-
Specify height: 1.5 `<-

Move the doorstop to touch the inside of the door. Use endpoint OSNAPs.
Hide `<-
Regen `<-
Check the solution layer to determine your accuracy.
Qsave `<-
**EXTRUDING POLYLINES INTO SOLID OBJECTS**
Creating a solid wall around and over the door opening and constructing banquette seating

Open the file: *extrude.dwg*
Saveas: *3d_extrude.dwg*
Current layer: *extrude*

---

**Extrude**  
*Creates unique solid shades by extruding an existing two-dimensional object*

- One object or a multitude of objects can be extruded (stretched as a solid) in the Z direction of the current UCS by defining the object and assigning an extrusion height.
- AutoCAD ignores a polyline’s width.
- A polyline must have at least 3 vertices to be extruded.
- Objects you can extrude: closed polylines, polygons, circles, ellipses, closed splines, donuts and regions.
- Objects you cannot extrude: crossing polylines, self-intersecting polylines, and objects contained within a block or nested block.
- An extrusion taper angle of zero will not taper the object in the Z direction. A taper angle greater than zero tapers in from the base object. A taper angle less than zero tapers out from the base object.
- The Path option will allow you to select a path or direction of the extrusion. The path should not lie on the same plane as the object(s) to be extruded, though one of the endpoints of the path should be on the plane of the profile.

Create a solid wall around and over the door opening by the counter.

Create a positive extrude with the polylines on the floor.  
**Extrude** <-  
- Select objects: *pick the 2 polylines on the floor*  
- Specify height of extrusion or [Path]: *13’3-1/2* <-  
- Specify angle of taper for extrusion <0>: <-

Create a polyline high above the door opening.  
**Pline** <-  
*Pick on the 4 endpoints of the newly extruded walls at the top*

Create a negative extrude for the polyline in the middle. The door is 6’8” high and the wall is 13’3-1/2” high so the resulting negative extrude is 6’7-1/2”.

**Extrude** <-  
- Select objects: *pick the polyline floating in air (#2)*  
- Specify height of extrusion or [Path]: *-6’7-1/2* <-  
- Specify angle of taper for extrusion <0>: <-
BANQUETTE SEATING

Rotate your UCS parallel to the side view.
Join the curving polyline with the 2 straight lines.
Then extrude it in a negative direction to create the banquette seating.

UCS, X, 90 <-

UCS, Y, -90 <-

Extrude <-
Select objects: pick the polyline you’ve just created
Select objects: <-
Specify height of extrusion or [Path]: -14’9 <-
Specify angle of taper for extrusion <0>: <-

UCS, world <-
Hide <-
Regen <-
Check the solution layer to determine your accuracy.
Qsave <-

REVOLVING POLYLINES INTO SOLID OBJECTS
Constructing a water glass and a pendant light

Open the file: revolve.dwg
Saveas: 3d_revolve.dwg
Current layer: revolve

Revolv e
Forms a solid by rotating a 2-dimensional object about an axis
- Objects that can be revolved include polylines, circles, ellipses, polygons, donuts and regions.
- Objects rotate around a defined axis of revolution. This axis may be created by selecting points on
  screen or with coordinates, or creating a line to represent the axis and using this as the object to revolve
  around.
- Objects rotate in the positive direction (clockwise).
- You cannot revolve blocks or objects contained within a block; use Explode to un-block an object or
  group of objects.
- You can revolve only one object at a time.

Get into the elevation view by moving your crosshairs to it.
Zoom closer to the table top and the glass profile.

Create a glass on top of the table.
The glass profile was created by drawing an enclosed polyline. A straight line was drawn close to it;
ortho was on. This line serves as the axis line of revolution.

Revolv e <-
Current wire frame density: ISOLINES=#
Select objects: pick the polyline that represents the profile of the glass
Select objects: <-
Specify start point for axis of revolution or
Define axis by [Object/X (axis)/Y (axis)]: with the endpoint OSNAP,
pick one end of the line
Specify endpoint of axis: with the endpoint OSNAP, pick the other end of the line
Specify angle of revolution <360>: <-

Zoom extents in all viewports <-
Create a pendant lamp

Put your crosshairs in the elevation view and change it to a right view.

*View, 3D Viewpoint, Right* 

**Zoom closer to the pendant light by the west wall.**

Create a lighting fixture by the west wall. The lighting fixture profile was created by drawing an enclosed polyline. A straight line was drawn close to it; ortho was on. This line serves as the axis line of revolution.

*Revolve* 

- Current wire frame density: ISOLINES=#
- Select objects: *pick the polyline that represents the profile of the pendant light fixture*
- Select objects: <-
- Specify start point for axis of revolution or
- Define axis by [Object/X (axis)/Y (axis)]: *with the endpoint OSNAP, pick one end of the line*
- Specify endpoint of axis: *with the endpoint OSNAP, pick the other end of the line*
- Specify angle of revolution <360>: <-

*Erase the axis of revolution line.*

Make 3 more copies of this light.

*Copy* 

- Select objects: *pick the pendant light*
- Select objects: <-
- Specify base point or displacement or[Multiple]: m <-
- Specify base point: 0,0 <-
- Specify second point of displacement or [use first point as displacement]: 0,3’ <-
- Specify second point of displacement or [use first point as displacement]: 0,12’6 <-
- Specify second point of displacement or [use first point as displacement]: 0,15’6 <-
- Specify second point of displacement or [use first point as displacement]: <-

*Hide* 

*Regen* 

Check the *solution* layer to determine your accuracy.

*Qsave* 

*Qsave*
**3Dorbit**  
*Interactive 3D viewing*

- By clicking and dragging with your mouse, you can twist, turn and spin a 3D model. An endless number of viewpoints is possible.
- You can manipulate any view, including plan, elevation, section, perspective, and isometric views.
- 3Dorbit displays on screen a circle divided into 4 quadrants. This is called an arcball.
- By picking on the 4 quadrant points or by picking outside the arcball, you can rotate the view around the X axis, the Y axis or the XY plane.
- You can view the entire drawing model or you can select particular objects within the model view.
- By right-clicking on the screen while in 3Dorbit, you can access any of the options listed below.

**Options:**
- **Pan**: Shifts the view in real-time without distortion.
- **Zoom**: Shifts the view in real-time with distortion.
- **Orbit**: Rotates the view around the arcball.
- **More**: Sets and adjusts the clipping planes plus other options.
- **Projection**: Rotates the view around parallel or perspective planes.
- **Shading models**: Hides and shades the model during rotation.
- **Visual aids**: Controls compass, grid and ucsicon settings.
- **Reset view**: Returns to the original view from the beginning of the command.
- **Preset views**: Picks one of 10 orthogonal or isometric views.

**NOTE:** Related 3D commands include: 3Dclip, 3Ddistance, 3Dpan, 3Dswivel and 3Dzoom.

View this 3D block Diagram using the 3D orbit command - it should look something like this:
Task [5]

3D EDITING
The task this week is to learn to construct and use 3D surfaces

slice   intersect   subtract
union

To complete this week’s task, you will need to:

- Make certain you have set enough time aside to complete the work,
- Place your completed work in the correct locations

The tools include

- The course pack instructions
- Movies
- AutoCAD help menu
- TA support on line and in labs
SLICING THROUGH 3D OBJECTS

Open the file: slice.dwg
Save as: 3d_slice.dwg
Current layer: cone

Turn off any active OSNAPs.

---

**Slice**  
*Cuts through solids with a plane*

- The sliced region or solid will be split into two parts. You can retain both sides or one desired side of the cutting plane. It is suggested that you retain both sides and erase the undesirable side later.
- The layer and color properties of the sliced solid are retained.
- 5 options:
  - Object Aligns the cutting plane with an existing polyline, circle, arc or ellipse
  - Zaxis The cutting plane is defined by specifying a point on the Z axis of the XY plane
  - View Aligns the cutting plane with the current viewpoint
  - 3points The cutting plane is defined by selecting 3 specific
  - XY/YZ/ZX Aligns the cutting plane with the current UCS X, Y, or Z direction

In order to slice through the cone above the counter, you need to tell the computer WHERE to slice. Create a box and move it up to the proper height. Use this box to pick the 3 points on your slicing plane. Then erase the box.

It's easiest to create the box in the plan view.

**Box**

- Specify corner of box or [Center]<0,0,0>: pick any point on the floor
- Specify corner of [Cube/Length]: L
- Specify Length: 12’
- Specify Width: 6’
- Specify Height: 1’

Move this box up to 6’8-1/2”. Use a base point coordinate of 0,0,0 and a second displacement point of 0,0,6’8-1/2.

Current OSNAP: endpoint

It's easiest to slice the cone in the isometric view, using the 3points option. Zoom closer to the box.

**Slice**

- Select objects: pick the cone above the counter
- Specify first point on slicing plane by [Object/Zaxis/View/XY/YZ/ZX/3points]
- Specify first point on plane: with the endp OSNAP, pick point #1
- Specify second point on plane: with the endp OSNAP, pick point #2
- Specify third point on plane: with the endp OSNAP, pick point #3
- Specify a point on desired side of the plane or [keep Both sides]: type B to retain both sides

**Erase** the tip of the cone. **Erase** the box.

Thaw layer: counter

Slice the cup you created on the countertop. Use the countertop itself as the slicing plane. Because this countertop is a solid, you cannot use the Object option to simply pick the counter.

Create a box and move it up to the proper height. Use this box to pick the 3 points on your slicing plane. Then erase the box.

**Box**

- Specify corner of box or [Center] <0,0,0>: pick any point on the floor close to the “cup”
- Specify corner of [Cube/Length]: /

---
Specify length: 1’ <-
Specify width: 1’ <-
Specify height: 6 <-

Move this box up to 4’9”. Use a base point coordinate of 0,0,0 and a second displacement point of 0,0,4’9.

Slice <-
Select objects: pick both cones on the counter
(One cone is inside the other - - really zoom in to find it)
Select objects: <-
Slicing plane by Object/Zaxis/View/XY/YZ/ZX/<3points>: 3p <-
1st point on plane: with the endp OSNAP, pick point #1
2nd point on plane: with the endp OSNAP, pick point #2
3rd point on plane: with the endp OSNAP, pick point #3
Both sides/<Point on desired side of the plane>: type B to retain both sides of the sliced solid

Erase the bottom of the 2 cones. Erase the box.

Hide <-
Regen <-
Check the solution layer to determine your accuracy.
Qsave <-

*The slice command was also used to cut the cylinders that form the countertops. Open your file 3d_cylinder.dwg and practice slicing on your own. Use the wall as the slicing plane to turn the circular counter into a semi-circular counter.

CREATING A NEW SOLID FROM INTERSECTING SOLIDS

Open the file: intersect.dwg
Saveas: 3d_intersect.dwg
Current layer: intersect

**Intersect**

*Creates a composite object from the intersection of two or more solids or regions*

- The order of selection is unimportant.
- The regions or solids may be in any plane.
- The composite solid or region includes the volume and mass enclosed by all the selected solids or regions.
- The intersect command will not work after issuing the hide, shade or render commands (issue the zoom or regen command to regenerate the drawing).
- Once two or more objects are “intersected”, they CANNOT be separated. You can a) use the undo command, or b) erase the composite object and begin again.
- The parts of the solids that disappeared when intersected CANNOT be restored without using the Undo command.
- If the solids or regions are on separate layers, the composite will be on the layer of the object selected FIRST. The color of the object, if separate and not controlled BYLAYER, will be maintained after issuing the union command.
Create a composite from the 2 boxes on the tabletop. Keep in mind that the intersect command will find the overlapping area of 2 or more solids and create a new solid from it. It will then erase the non-overlapping parts of the solids selected. This is unlike the interference command, which creates an additional solid where 2 or more solids overlap.

*Intersect* <-
   Select objects: *pick the 2 boxes*
   Select objects: <-

The computer will calculate where these 2 boxes overlap and will do a “double subtract”. The parts of the solids that did not overlap will disappear. You cannot get these back without using the Undo command.

Create another composite from the wall in the SW corner. This is the same corner on which you used the interference command earlier. See the difference between the two commands.

*Intersect* <-
   Select objects: *pick the 2 walls*
   Select objects: <-

*Hide* <-
*Regen* <-

Check the *solution* layer to determine your accuracy. *Qsave* <-
Using AutoCAD in Your Job: Part Two

SUBTRACTING ONE SOLID FROM ANOTHER

Open the file: subtract.dwg
Saveas: 3d_subtract.dwg
Current layer: cone

**Subtract** 
Joins regions or solids by subtraction

- The Subtract command forms a composite which contains the area and volume of two or more solid objects combined together by SUBTRACTION.
- Objects selected can lie on the same or different planes.
- The first object selected will dictate the layer and color of the composite.
- The subtract command CANNOT be reversed once a drawing has been saved. The solids can either be exploded back to their original wireframe representations, or erased completely.

There are 2 cylinders and a sliced cone above the counter. Subtract the inner cylinder from the outer cylinder to form a soffit for the recessed lighting. It might be easiest to make your selections in the isometric view.

**Subtract**

Select solids and regions to subtract from...
Select object: select the OUTER cylinder
Select objects: <-
Select solids and regions to subtract...
Select objects: select the INNER cylinder
Select objects:

Thaw layer: counter
Current layer: counter
Freeze layer: cone
Finish the counter design by subtracting the inner cylinder from the outer cylinder. This will open up space for an employee to stand and serve customers.

Move your crosshairs into the plan view viewport. It will be easiest to select the cylinders in this view.

**Subtract**

Select solids and regions to subtract from...
Select object: select the OUTER cylinder (#1)
Select objects: <-
Select solids and regions to subtract...
Select objects: select the INNER cylinder (#2)
Select objects: <-

Hide <-
Regen <-
Check the solution layer to determine your accuracy.
Qsave <-
COMBINING MULTIPLE SOLIDS INTO ONE SOLID

Open the file: union.dwg
Saveas: 3d_union.dwg
Current layer: cone

Union

Joins regions or solids by addition

- The Union command forms a composite which contains the area and volume of two or more solid objects combined together by ADDITION.
- Objects selected can lie on the same or different planes.
- The first object selected will dictate the layer and color of the composite.
- The union command CANNOT be reversed once a drawing has been saved. The solids can either be exploded back to their original wireframe representations, erased completely, or you can issue the Undo command.

Combine the cylinder and cone above the counter.
Union <-
   Select objects: select the two solids above the counter
   Select objects: <-

Zoom closer to the back wall (the purple wall layer).

Combine the three parts of the wall. These are the same parts you extruded earlier. Right now they are separate pieces, but they should be one fluid solid object.

Union <-
   Select objects: select the 3 solids that form the wall
   Select objects: <-

Hide <-
Regen <-
Check the solution layer to determine your accuracy.
Qsave <-
AutoCAD REVIEW OF COMMANDS & CONCEPTS

It is to your benefit to review the concepts and commands associated with moving objects in 3D space and presenting them in paper space.

You will be using these commands throughout the rest of the semester

Coordinates

3 ways to enter information based on the Cartesian Coordinate System

• Absolute:  x,y,z
• Relative:  @x,y
• Polar:  @length<angle

Copy in 3D  Creates duplicate objects in different planes

Move in 3D  Alters the location of selected objects in different planes

• Remember how to...use the origin (0,0,0) as the basepoint and alter the necessary coordinate as the second point of displacement (x,y,z)?

Dist  Calculates the distance and direction between two points

• Remember how to...get the distance and direction between two selected points?

Ucs  Controls all available User Coordinate Systems

• Remember how to...restore saved coordinate system locations and alter the location of the UCS?

DDvpoint  Specifies the direction to view the model

• Remember how to...alter the view of the model through the pull-down menu preset options (under View)?

List  Displays information about selected objects

• Remember how to...display information about any object you select?

Mspace  Switches from paper space to a model space viewport

• Remember how to...switch to the model while on the papercspace presentation sheet?

Mview  Creates and manipulates viewports in papercspace

• Remember how to...create new viewing windows for your presentation?

Plan  Displays the plan view

• Remember how to...display the drawing flat on the floor?

Pspace  Switches from a model space viewport to paper space

• Remember how to...switch to the paper while on the papercspace presentation sheet?

Tilemode  Passage from model space to paper space

• Remember how to...toggle between the model space side of the computer versus the paper space side?

View  Saves, restores and manipulates existing views

• Remember how to...save and restore interesting views?

Vports  Splits the display area into tiled viewports

• Remember how to...have a single viewport or multiple viewports?

Dview  Creates parallel projections and perspectives

• Remember how to...create perspective, elevation and section views?
Task [6]

Exhibit in 3D solids
The task this week
Construct in 3D solids, the exhibit you selected in Task 2

To complete this week’s task you will need to:
Set aside time to listen to the instructions on the movie
Place your finished work in the appropriate files

The tools include
movie
TA lab and on-line support
Task scan sheet
Task [7]

Foundation, Floor, Wall
The task this week

Assemble a structure

To complete this task you will need to

Listen to the instruction on the movie
Assemble the structure using the 3D solids provided as blocks in the drawing file
Place the completed work in the appropriate locations

The tools

Movie
Foundation_Floor_Wall drawing file
TA support in labs and on line
STRUCTURAL DEFINITIONS

General important structural terms and concepts

**Force**
Something acting from the outside that tends to change the state of rest or motion

**Stress**
Internal resistance to an external force

**Resultant**
Resistance necessary to 0 a force

**Deflection**
Amount of space that a material bends and still returns to its original shape

**Elasticity**
Ability of an object to maintain its shape

**Brittle**
When a material has little/no elasticity

**Elastic limit**
The point where a material has been stretched beyond the point it can return

**Yield Point**
Point at which an object breaks

**Buckling**
When a material bends under excess compression

**Gravity**
Atmospheric pressure

**Center of Gravity**
Center of weight of an object

**Racking**
Unstable construction; wiggling

**Load**
Something external weight

- Point load - all weight applied at one point
- Distributed load - weight is spread out
- Live load - things that are active, like people, furniture & weather
- Dead load - the building itself; the structure’s weight

**Torsion**
Stress produced in an object by twisting along its longitudinal axis

**Plasticity**
Condition where material will begin to deform when weight of the load becomes greater than that material’s elastic ability

**Substructure**
Below earth

**Footing**
Converts downward pressure out distributing the load

**Foundation Wall**
Supports load of building above ground line; transmits weight of building to footing

**Superstructure**
Above earth

**Subflooring**
Rough flooring under finish floor that rests on floor joists

Terminology generally associated with wood or steel construction

**Joists**
Horizontal structural member that supports floor system

**Girder**
Horizontal beam supporting floor joists

**Sill Plate**
2 x 6s placed directly atop foundation wall

**Sole Plate**
Horizontal framing member directly under studs

**Stud**
Upright beams in framework; usually 2 by 4s and spaced at 16’ on center (oc)

**Header**
Piece of lumber supporting end of joist

**Top Plate**
Goes around top perimeter of framework

**Rafter**
Structural member used to frame a roof

**Cantilever**
A horizontal structural member, a beam, a girder, or a truss, that projects from a supporting column or wall

**Truss**
Prefabricated triangular-shaped unit for supporting roof loads over long spans

Terminology generally associated with wood construction

**Nominal Dimension**
Actual finished size of limber (ex: 2 by 6 is really 1.5 by 5.5)

**Plywood**
Thin sheets of wood glued in crossgrain

**Composite Board**
Wood particles and adhesive; cheaper than plywood

**Woolminized wood**
Wood treated with an oil product to make it rot and insect resistant

**Spline**
Piece of wood put into joint to generate more surface

**Toenailing**
Crossing nails through stud into sole plate; can not be pulled apart easily

Terminology generally associated with metal construction

**Welding**
Heating metal enough so that additional material can penetrate it
<table>
<thead>
<tr>
<th>Term</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Brazing</td>
<td>Similar to welding but metal is not penetrated</td>
</tr>
<tr>
<td>Soldering</td>
<td>Similar to welding but metal is not penetrated</td>
</tr>
<tr>
<td>Anodizing</td>
<td>Process of applying a dye that penetrates the outside layer of aluminum</td>
</tr>
<tr>
<td>Plating</td>
<td>A finish put on metal</td>
</tr>
</tbody>
</table>

**Other important miscellaneous terminology**

- **Capillary Action**: Ability of water to move from one side of material to the other
- **Jig**: A temporary construction to hold pieces in place while they are being secured
- **Knocked Down**: Furniture not assembled
- **Safety Factor**: The increased ratio of something to be extra safe
Task [8]

Window, Door, Baseboard, Roof
The task this week
Assemble a structure

To complete this task you will need to
Listen to the instruction on the movie
Assemble the structure using the 3D solids provided as blocks in the drawing file
Place the completed work in the appropriate locations

The tools
Movie
Window_Door_Baseboard drawing file
TA support in labs and on line
PRINCIPLES OF STRUCTURES

Here are some very important structural concepts.
Some time the structure will dictate the design and sometime the design will dictate structure.
No matter which way it goes, you cannot design for one without regard to the other.

**Compression**
- An external force that tends to shorten the members of the structure by pressing or squeezing them together
- An internal resistance to that external force
- Opposite of tension

Good materials for resisting compression forces include:
- Brick
- Tile
- Steel

**Tension**
- An external force that tends to pull outward on the members of a structure, stretching or lengthening them
- A internal resistance to that external force.
- Opposite of compression

Good materials for resisting tension forces include:
- Cable
- Chain
- Steel

**Shear**
- An external force acting on adjacent layers of a material tending to cause them to slide past one another.
- The shear force is defined by the direction of the force. There are typically two common shears

**Horizontal Shear** as in a beam

**Vertical Shear** as in a column
**Bending Moment**

- A moment is defined as the tendency of a force to cause rotation about a given point of axis

When you talk about a bending moment, you are talking about a lever arm and its ability to multiply force dramatically. This lever action appears multiple times in structural systems. When you become aware of it, you will be amazed how often you will see it.

**TRIANGULATION**

- The three point geometry of the triangle is the most basic key to providing stability.
- Stability ALWAYS involves triangulation.

**STRUCTURAL SYSTEMS**

The structural system is made up of structural parts and connectors which act together. The system becomes greater than any one of its parts.

Ching's text puts things into 3 traditional types of structural systems. There are really only 2 types with the 3rd type being a combination of the first 2.

**Linear System**

- This is often referred to as the post and beam system.
- The most common building system
- It is made up of load bearing columns (vertical supports) and beams (horizontal supports)
- A frame (skeleton) is constructed and then walls (skin) are attached
- Curtain wall or skin is a wall that is supported by the frame and is non structural

**Planar System**

- Made up of, load-bearing planes (walls) that support the structure
- Walls can not be removed without compromising the integrity of the structure

**Composite System**

- The composite system uses both the linear and the planar systems.

Study Ching and learn the names of the components that make up the various structural system here are some other structural systems to consider. Use them to spark your imagination.
Let's keep this simple. All the structural systems must carry and transfer load. That's the purpose of a structural system. All loads are transferred one of two ways, as either:

A **Point load** - where the load is transferred through the beams to the columns to the base. (Linear system)

The advantage is that the structural frame provides tremendous flexibility. Nothing else in the structure needs to be considered structural - everything is attached to and supported by the structural frame.

The disadvantage is that all the load is transferred to a very small surface and is therefore the impact on that small base is very severe.

A **Distributed load** - where the load is spread over a large surface like an entire roof or wall. (Planar system)

The advantage is that the impact of the load on the base is diminished.

The disadvantage is that each wall or panel is critical to the integrity of the structure. Any changes in a panel can impact the entire structure.
Task [9]

Directives & Concepts
The task this week

Begin to modify and improve the exhibit you worked on earlier
Develop a design directive
Develop a concept statement

To complete this task you will need to

Listen to the instruction on the movie
Take time to develop and revise the statements
Develop multiple directive and concept statements to construct 1 good one
Place the completed work in the appropriate locations

The tools

Movie
TA support in labs and online
THE DESIGN DIRECTIVE

A Directive is an order, command, instruction, or goal - a statement of WHAT has to be done

A Business Directive is a business goal. In its broadest terms it is to provide a service and generate a profit. It is the corporate executives' responsibility to determine WHAT from a business perspective, they are going to do to achieve that goal

In the article, Starbucks executives identified directives or goals they believe will answer the need of the market place and thereby generate a profit for the company.

Not all the goals that were identified can be communicated through design. For example; you cannot convey 'to create a total experience' with only the design, since the design is only a part of the total experience.

Some of the goals can be achieved more easily than others. Some goals will be more effective than others.

A Design Directive is a design goal. Its an order by the company of EXACTLY WHAT the design is expected to do and to say.

The Design Directive comes directly from the list of Business Directives. It is the designer's responsibility to communicate the Directive through design.

In this project YOU are to develop a design directive statement relating to the exhibit you choose in task 2

Choosing the right Design Directive is your first step in the success or failure of the project.

Sometimes, the design directive is very specific and limiting; at other times, it is very broad and vague. The more specific and limiting the directive, the easier it will be to recognize its success or failure. The broader the directive, the more latitude the designer has and the greater the risk of missing the mark or being misunderstood. BE SMART AND CHOOSE ONE THAT IS SPECIFIC! This will require work

Every final design must:

1. CLEARLY CONVEY THE DIRECTIVE IN THE DESIGN
2. BE UNIQUE AMONG THE COMPETITION
3. ELICIT AN EMOTIONAL RESPONSE
4. BE SIMPLE & EASY TO UNDERSTAND
A DESIGN CONCEPT - WHAT IS IT?

A Concept is an idea or plan of action. The concept answer the question of HOW the WHAT (directive) will be accomplished.

There are Business Concepts

Business directive:
Reduce the cost of doing business (Chrysler Corp) WHAT
Business Concept:
Terminate the design employees and hire them back as independents - (This removed the costly benefits package) HOW

Business directive:
Reduce the labor and increase production (McDonalds Corp) WHAT
Business Concept:
Each employee is a specialist producing a part of the meal rather than producing the entire meal HOW

There are Design Concepts

Design directive:
Maintain an association with the ‘beetle’ (VW Corp) WHAT
Design Concept:
Maintain a similar contour as the original HOW

Design Directive:
Reduce the long lines at the bank (Banking) WHAT
Design Concept:
Make several short lines with ATM machines HOW

A design is built on a Design Concept. The design concept is the designer’s idea or plan for manipulating space and applying meaning that communicates and carries out the primary corporate directive. It is always the answer to HOW it will be done. It has NOTHING to do with what it will look like.

“Concept is the most important factor in the way we design. Properly developed at the outset of each project, it guides you and helps all the details fall into place as you move toward your goals.” (Interiors, January 1997)

Carolyn Iu, AIA
Interior Design Hall of Fame

A SUCCESSFUL DESIGN CONCEPT

Successful concepts include the following characteristics:

1. The concept must have an obvious link to the Business Goal. The concept must be able to be understood and embraced by the business decision makers.

2. The concept should be a UNIQUE method for meeting the design directive. It is your profession to generate design solutions that are innovative, and unique

3. The concept must be exciting - emotionally charged.

4. The concept must be visible and visually understood from the design solution. If people don’t see it or understand it from the design, you missed it!

The concepts are presented in the form of a simple, easy-to-understand word picture - - that a CEO can take with him anywhere

ie. We are going to put a man on the moon using a sling shot. If approved, the designer will begin designing how that slingshot will work and what it will look like.

A successful concept is simply a new and better way of doing something. When we see it, we say - - why didn’t I think of that?
DESIGN RALLY - One method of brain-storming

A design rally is one of many brainstorming techniques used by designers to generate design ideas. It's a good technique for getting you started. The idea is to loosen up and allow others to assist you in generating ideas while you assist them in return.

Although designers do generate original ideas, they are also skilled in recognizing good ideas, modifying them as necessary, and using them. This makes sense when you consider that creativity is simply making a new association.

The design rally takes advantage of the fact that we always find it easier to solve someone else’s problem than it is to solve our own, since we have such high emotional involvement and critical attitude toward own projects - - we often miss the big picture as well as the frivolity of life that makes up good design. The design is to be enjoyed, and if you are having a good time developing the design, the design always shows it. If you are not enjoying the project, the result is equally obvious.

Decide to have fun with this project and the person reviewing it in your portfolio will enjoy it too!

We will conduct the design rally in class. Bring 8.5 x 11 sheets of paper with you to this class session. No instruction here. The rally involves spontaneous thinking, so I can’t give you anything in advance.

You do not need to participate in the design rally to finish this assignment, but you would be very foolish to miss it.

DEVELOPING ADDITIONAL DESIGN IDEAS

Here are 4 ideas for generating additional ideas:

1. Get as much information about everything associated with this project.
   You will need to have data in order to have something to rearrange.
   You need to get past the obvious to get to the obscure - - the areas others have missed.
   Become obsessed with the project - - to the point of frustration.
   Look for both problems and solutions

2. Continue the design rally on your own with a friend
   Ask questions: Why do people drink coffee? Why at a kiosk in a mall?

3. Look at everything from the reverse point of view - take something that is usually small and make it big. Take some thing that usually happens first and make it the last.

4. Force new associations using both vertical/lateral thinking.
WRITING A DESIGN CONCEPT STATEMENT

The concept is best presented in a single sentence and/or simple diagram which lends itself to discussion with the corporate decision makers.

The best concept statements are immediately understood word pictures that can be easily remembered and repeated.

Here’s an example: “The space shuttle will land like a glider”

DESIGN CONCEPT MATRIX

Develop 6 design ideas (concepts) for achieving the design directive.
Feel free to use ideas that were generated during the design rally.
State the concept with as few words as possible.
State the concepts that will obviously achieve the directive.
State the concepts that will be VISIBLE in your final design.
These are HOW statements.
i.e. The design will . . . . . . . . by . . . . . . . .

1. ________________________________________________________________________
2. ________________________________________________________________________
3. ________________________________________________________________________
4. ________________________________________________________________________
5. ________________________________________________________________________
6. ________________________________________________________________________

Now you need to identify the Design Concept that has the most potential.
Use an evaluation chart.
We begin by listing things the design concept Must do.

We list them in their order of importance.

It Must:
1. The concept must address the design directive
2. The concept must be visible and visually understood in the design solution
3. The concept must be able to accomplish the goal within space limitations

We follow this list with a list of things that we personally Want the design to do.
Again, we list them in order of importance.

I Want:
1. The concept to be unique
2. The concept to be emotionally charged
3. The concept to be simple
To score each of the ideas:

Decide if your first idea meets the first criteria listed under the first heading ‘The design MUST’. If yes, enter an ‘X’. If no, leave it blank. Compare the first idea with each of the remaining criteria following the same method.

Enter in the ‘Total’ column, the number of ‘Xs’ that idea 1 received - counting from left to right until the first blank is encountered. Once you hit a blank, stop counting. Repeat the process for each of the remaining solutions.

The solution with the highest score is your best option - highlight it.

Table 1: DESIGN CONCEPT MATRIX

<table>
<thead>
<tr>
<th>IT MUST...</th>
<th>1. The exhibit will be modular</th>
<th>X</th>
<th>X</th>
<th>X</th>
<th>X</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3.</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4.</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5.</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6.</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7.</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
5. COMPLETING YOUR KIOSK DESIGN

Keep in mind that your design will be first evaluated based on its creativity and ability to meet the criteria of the directive. (remember: CEOs make hiring decision in the first 30 sec)

There must be an obvious connection. The Concept is evaluated in relation to the Design Directive. The Image is evaluated in relation to the Concept.

The kiosk should be workable but that is not it’s most important criteria. The critical path is its innovation and its ability to present a new direction.

REVIEW ALL THE COMPETING PRELIMINARY SUBMISSIONS - KNOW YOUR COMPETITION
Continue to refine your design. You have (of course) the opportunity to change your design based on your preliminary design critique (during lecture time) but this is NOT the time to take a radically new approach.
Task [10]

Exhibit preliminary
The task this week
- Develop an image for the exhibit
- Make a preliminary presentation of your exhibit
- Present your design directive
- Present your concept statement

To complete this task you will need to
- Listen to the instruction on the movie
- Take time to further develop and revise the directive and concept statements
- Place the completed work in the appropriate locations

The tools
- Movie
- TA support in labs and online
Before we begin working on the preliminary we need to talk about image

WHAT YOU NEED TO KNOW ABOUT IMAGE

The image IS the message.
Your task is to create the image that communicates the message of the design directive.

Here’s how it works.
All images trigger recall. They effect both the intellect and the emotions.
To communicate a message via the language of design.

There are 3 possible options:

1. In the first scenario, the viewer is presented an image they have never seen before.

   The viewer is taught/told what we want them to see.

   The viewer sees what we want them to see.

   For this to work, the explanation must precede the introduction of the image. Only then will the viewer see what we want them to see and the design will take on the meaning we want it to mean.

2. In the second scenario, the viewer is presented an image they have never seen before.

   The viewer is not taught/told what we want them to see.

   The viewer immediately relates the image to similar objects in their past and assigns that meaning to the object.

   Here is an example - - The introduction of the 1997 Corvette. GM assumes we are familiar with the contours of the F series fighters - - so we will transfer our association with the fighter aircraft to the automobile. GM wanted to associate the corvette speed with the speed of the fighter.

3. In the third scenario, the viewer is presented with an image they are familiar with.

   Most viewers will associate the kings crown with royalty & luxury.
FIND 6 IMAGES THAT COULD COMMUNICATE YOUR DIRECTIVE

Before you select your images, let's look at some examples.

How do you make something appear to be more valuable than it really is?

Look at what Tiffany does - www.tiffany.com. Tiffany visually separates the object from any background. Very simple and very elegant. It suggests the idea that there is no competition.

Look at what Harry & David do - www.harryanddavid.com. Harry & David take a good piece of fruit and increase its value by packaging it like an expensive piece of candy wrapped in gold. The fruit is gently held in tissue until you receive it. It's treated as if each piece was worth $100.00. The package is likely more costly than the fruit.

An example of solving the same problem with completely different outcomes?

Which of the two chair would you select?

Association is used to communicate the message.

The victorian chair is an expression of an individually crafted piece of sculpture. A throw-back to times past when time allowed for more to create a work of art.

The Barcelona chair is a reaction to the victorian style - it’s the introduction of machine produced sculpture. Wood is replaced with chrome-plated steel and velvet is replaced with leather. Spring seat construction is replaced with straps. One design retains the past while the other expresses the future.

The problem is the same. The result is substantially different because the concept behind each chair is substantially different. The image reinforces the concept.

This is the kind of change you need to initiate in the kiosk project. The same problem needs a new
What happens when we have the same concept but different furniture?

Here we have an example of two offices with workstations by two different manufacturers. www.hermanmiller.com and www.steelcase.com.

The furniture are different and the companies are different but they look very similar. Why? Because the concept is the same/similar.

Until someone comes up with a new concept for office design, all offices with workstations will continue to appear very much alike.

You are going to have to do better than just change the materials. You are going to have to come up with a new concept for the coffee business.

If you are having trouble selecting an image, the problem is probably the result of a weak concept.

IDENTIFY THE UNIQUE CHARACTERISTIC OF EACH IMAGE

Once you find the image you want to communicate, you need to identify the associations that make that image unique. Those are the key components to include in your design.

You do it this way. Let’s first go back to the jet fighter and the Corvette C5. What’s the concept that has generated the image in both the fighter and the corvette? It’s called slip-stream design. Its a wind tunnel tested geometry that reduces drag which in turn increases speed and reduces fuel consumption. That’s why the look is similar. They are both designed with the same concept.

You need to find the concept for your kiosk image?
Let’s say you like the idea of having coffee when you are on a picnic - OK - What is it that makes a picnic a picnic? It’s the outdoors, in the country, under trees, on the grass, in fresh air, etc. If you include or suggest these elements, you can create the picnic.

What makes a French Cafe a French Cafe? It’s outdoors, in a city, with small tables, awnings, umbrellas, etc. If you include or suggest these elements, you can create the french cafe.

The image needs to re-inforce the concept.
Write out your image statement

Example: the exhibit will remind you of:

To incorporate the image in your design, you need to have:

- Identify the key components that define that image.
  (i.e. in the French Cafe - the key components included
  the umbrellas and the small tables.)

- Developed 3D solid abstractions of those components

  Keep it simple. An umbrella can be constructed
  from a sliced sphere & a cylinder

  A table can be constructed using 2 cylinders
PRELIMINARY KIOSK DESIGN

A preliminary design is an abstraction of the final design. It describes the final design and includes only enough detail for the client to visualize the potential final outcome.

The task for the designer is to bring together all the components into a single design and then present that design to the client for further direction and clarification. The purpose for the presentation is not to show your design as much as it is to get further direction by either approval or disapproval and redirection.

Here is what the design should already include:

DESIGN DIRECTIVE   ... to express quality

CONCEPT   To have the exhibit made up of antique furniture instead of typical counters cabinets, etc.

IMAGE   The package is better than the product.

You have a series of unrelated parts (nothing fits).

You are going to modify your 3D solids drawing to meet the need of your directive, concept, and image.

The reason you have done each of these previous tasks independently is so that you will know what you are compromising when you begin fitting the parts together. The idea is to let the design evolve by beginning with the concept, and adding each of the independent components in order.

Each time you add a component, the entire design will need to be adjusted. That's to be expected. Let the design evolve WITHOUT COMPROMISING THE CONCEPT. Compromise something else.

Since the emphasis of this task is on creativity, emotional response, and innovation, the concept is the most critical factor to leave uncompromised.

Anything and everything else may need to be modified in order to optimally present the concept.

The preliminary design is an abstraction leading to the final design. It suggests the final design and includes only enough detail for the designer and the client to visualize and discuss the potential final outcome.

The preliminary design is made up of a series of area boxes located in 3 dimensional space. Each area box encloses equipment. Their size and shape is defined by the equipment they contain. The preliminary design presents these group boxes and NOT the equipment contained in them. You drew the equipment earlier to define the size.

Here is a simple example of what it might look like:
The preliminary design drawing should include the following information:

- **PLAN VIEW** - freeze the mview border
do not use the hide command
scale is 1/2” = 1’0”

- **ISOMETRIC VIEW** - freeze the mview border
do not use the hide command
you determine the best viewpoint;
no scale

- **PERSPECTIVE VIEW** - use DVIEW to construct your perspective
keep your eye level at about 5’
no scale

- **DETAIL** - Insert into this quadrant the image photo you selected

- **DRAWING LABELS** - make sure the scales listed in the drawing labels reflect the scales of the drawings displayed; do not move the locations of the drawing labels (these will be consistent with the cabinet template file you will use later in the course)

- **NOTES** - list your design directive, your design concept statement, and your image statement

  **IF YOU DO NOT LIST YOUR 3 STATEMENTS YOUR PROJECT CANNOT AND WILL NOT BE EVALUATED**

  list any additional notes describing anything not apparent in your design;
  use a simplex font at 3/16” height

- **IMAGES** - reload the msulogo in the titleblock if necessary by using the IMAGE command;

**PREPARING A PRELIMINARY PRESENTATION**

The purpose of a presentation is to present the design ideas to the client for discussion.

The success of the design depends on its ability to communicate the concept in the form of a 3D image.

This drawing should describe general sizes and shapes plus the arrangement of work functions. In addition, a clear linear progression from the **design directive** to the **design concept statement** and, finally, to the **image statement**.

Feedback is needed at this time to ensure that you are on the right track, before you spend your time detailing the drawing.
Task [11]

Exhibit FINAL
The task this week
   Render the exhibit
   Insert people
   Insert background
   Final presentation of your exhibit

To complete this task you will need to
   Listen to the instruction on the movie
   Take time to further develop and revise the directive and concept statements
   Place the completed work in the appropriate locations

The tools
   Movies
   TA support in labs and on line
**AutoCAD RENDERING COMMANDS**

**NOTES ON RENDERING:**
The process of creating color, shading, and pattern is substantially different using computer media than it is for traditional media processes.

In traditional media you are familiar with mixing pigments. Pigments are described using their Hue (color name), Lightness (value), and Saturation (purity) characteristics.

Your monitor is a source of light that emanates color rather than reflecting it. You are not seeing pigment color but light color. Mixing light colors uses the additive system of color.

The primary light colors are red, green, and blue. This is the reason computer color systems are called "RGB" systems.

This is important to remember when you are generating your rendering because you generate it using the light color system and you print your rendering using the pigment color system. That's why the results may appear different on the screen than they do on the printed page. What's more, color often varies from monitor to monitor and printer to printer.

**NOTE:**
1. A bad design rendering is just a bad design with color and texture. Work out the problems of your image statement and kiosk relationship before you render it.
2. Rendering is intuitive and an art. Not everyone agrees on colors and textures.
3. Materials are more than colors. They have pattern and texture.
4. If you don’t attach a material to an object, that object will render to the color of the layer.

Open your kiosk model presentation (the one with the template). Render the kiosk, using ACAD’s existing material library.

(Open your kiosk model presentation (the one with the template). Render the kiosk, using ACAD’s existing material library. Change the color of your AutoCAD graphics screen from black to white)

*Options* <-
Pick on the Display tab
Push the button Colors...
Color Options: Pick the arrow next to Color and select *White*
Push Apply & Close
Push OK

**Rmat**
Organizes rendering materials

- 3D objects in your drawing will be rendered, depending on the materials associated with each object.
- You can choose to have the software calculate the materials by the color of the layer or you can attach different materials independently to pieces of your drawing.
- You can create your own materials (standard, wood, granite, and marble) or use ACAD’s material library.
- Only 3D solid objects can be rendered.
Rmat -
On the left of this box is a listing of the materials currently available in your drawing. (It's probably empty now).

To add materials to this list, push the button Materials Library...
In the Materials Library dialog box, scroll down the listing on the right.

When you come to a material you would like to preview, highlight it and push the preview button.

When you find a material you'd like to use, add it to your Materials List by pushing the <- Import button in the middle of this box.

When done, push OK.

To attach a material to a specific object in your drawing, highlight the material in the Materials List on the left and push the Attach< button.
Select objects to attach "???" to: pick the desired object in the drawing
Select objects: <-

When returned to the Materials dialog box, continue importing and attaching materials as needed for your design.

When you are done, push OK

When you would like to see the objects rendered, use the Render command.

To detach a material - highlight the material from the listing on the left and push the Detach button. The software will search through your drawing for objects with this material and detach it from those objects.

Get inside the perspective mview. Since this is the only view with rendered materials, don’t spend your time rendering something that’s not going to be visible in the perspective view (i.e. something under the counter, behind a wall, etc.).

**Render**

Creates a realistically shaded image of a 3D solid model

- Render provides the most realistic visual presentation of your 3D model.
- The current scene and light settings can be used as defaults to produce a “Quick Render”.
- The render box includes settings for Rendering Type, Rendering Procedures, Rendering Options, Destination, Sub Sampling, Scene to render, Background and Fog/Depth Cue.
Render

Confirm that the Rendering Type: is Photo Real
Confirm that the Destination: is Render Window
Confirm that there is a checkmark by Shadows under Rendering Options
Push Render

The Render Window will appear in front of your graphics screen (see illustration at right). This window cannot be closed; it will disappear when you exit the software.

When the kiosk perspective is rendered to your satisfaction:

In the ACAD Render Window:
Under File, select Save.

In the Save BMP dialog box, give the file a name (the default will be the name of your current drawing file);

Save file as type: BMPs (*.bmp).

On the right side of this box, find and select either your personal AFS site (P:drive) or your floppy disk (A:drive).

When you are done, switch to your graphics screen by picking the ACAD icon in the taskbar at the bottom of your screen.
RENDERING YOUR DESIGN

Return to your exhibit presentation drawing.

Tilemode to paper space.

*Erase the mview for your perspective.* Do NOT erase the perspective label. You will replace this perspective with your rendered image.

*Image, attach,* find and select your rendered image file (*drawing name.bmp*).

**Do not use DDINSERT or XREF!**

At: *place the BMP file visually above the perspective label*

Scale factor: *select a scale factor greater than one; if too small, repeat process until size of image is adequate*

Rotation angle: 0

*Imageframe,* turn it off.

Adjust your other views as needed.

Freeze your mview borders.

Qsave <-

INCLUDING PEOPLE IN YOUR DESIGN

Never forget, you are designing for people, if it doesn’t fit the people it’s not successful

To include people in your design is VERY important - not just one or 2 but several
People will immediately give the viewer a sense of scale.
The images of the people in relation to the objects will tell you if the counter height is correct,
The images of the people will tell you if your clearances are correct

The movie shows you exactly how to do it

INCLUDING A BACKGROUND

The background is VERY important

It’s the choice of background that tells you if the exhibit is indoors or out doors

*It’s the other exhibits that become the measure of your success - they will make you or break you what looks good in one context may not in another*

You select the background and the movie shows you exactly how to do it
TIPS FOR A SUCCESSFUL FINAL PRESENTATION

Let's talk about presentation - - not your design - - but the presentation of your design.

MAKE SURE THE SHEET IS FULL
The most common mistake I see is that the student's drawings lack enough drawing. Here's what I mean. The professional has so much to say about their design and its construction that paper becomes a premium. It's full of drawings and notes. The student, on the other hand, doesn't have enough to say to begin to fill the sheet. It's the difference between a cartoon character and a portrait. Make sure you have enough drawing and have included notes.

ESTABLISH A HIERARCHY OF VIEWING
Another common mistake is to complete all of your drawing to the same level of detail. Whether everything is highly detailed or not detailed at all, the situation is the same - - the eye doesn't know what you want it to attend to, so it wanders aimlessly over the drawing.

Here's the solution:
Identify the most important aspect of your design and really detail it. Make it very believable. This will buy you two things;

a. It will draw attention right where you want it.

b. It will make the rest of the drawing more believable.

Back to the illustration of the difference between a cartoon and a portrait......
When you try to draw from memory or try to make up something, you will find yourself making a drawing that looks like a cartoon. To draw a portrait, you have to have the person or at least a photo of that person in front of you. EXACTLY the same thing happens here. You will have to have a photo in front of you by your computer when you draw the object.

MAKE SURE YOU TELL THE CLIENT WHAT YOU WANT THEM TO SEE
Make sure you include NOTES. Include the Directive, the concept, and the image. This will direct the viewer's attention to see what you want them to see, and it will add to the completeness of the drawing.

One word of caution: Don't make the text style too large. Too large a font size also makes it look like student work. Keep the text under 1/8" high on a 24x36 drawing.

One last word about your notes - - do a spell check. It's easy to miss typos. The people to whom you are presenting are very good at picking up spelling errors. Save yourself some embarrassment.

This final presentation is an expansion of the preliminary presentation.
A rendered perspective view is required.

It's the amount of detail that makes the difference between your preliminary design and the final design!
Task [12]

Drawing a Cabinet
The task this week

Draw in 3D solids, a cabinet from your residence

To complete this task you will need to

- Listen to the instruction on the movie
- Take time to further develop and revise the directive and concept statements
- Place the completed work in the appropriate locations

The tools

- Movies
- TA support in labs and on line
UNDERSTANDING THE CONSTRUCTION OF YOUR CABINET

DON'T EVEN THINK ABOUT BEGINNING TO DRAW A CABINET WITHOUT HAVING THE CABINET IN FRONT OF YOU! YOU HAVE TO SEE IT, MEASURE EACH PART, AND UNDERSTAND THE CONSTRUCTION TO DRAW IT.

There are 3 major approaches to the construction of cabinetry

1 CONSTRUCT THE CABINETRY ON THE SITE
   This is done when the piece of cabinetry is too large to transport
   This is the MOST expensive. The cabinet maker needs to set up his shop on site.
   (This is both a big expense and a big hassle)
   Truely a one-of-a-kind piece

2 FACTORY BUILT CABINETRY
   All the equipment is already there at the factory
   The cabinet has been produced many times (mass produced)
   The cabinet is simply dropped off at the site and set up.

3 FACTORY BUILT CABINETRY WITH CUSTOMIZATION
   Best of both worlds
   Cabinets are set up and the cabinet makers make the necessary modifications

The approaches to the construction of cabinetry are very similar to the approaches used to design the structure. The two most common approaches include;

1 A framed cabinet
   Design a frame and fill it

2 A frameless cabinet
   Construct the cabinet from panels

A constantly reoccuring decision is deciding which panel overlaps which panel. You will find that you cannot construct a 2' x 4' cabinet using 3/4" plywood with 2 front panels each 4' in length and 2 side panels each 2' in length. When you assemble them, the length may be 4' but the width will be 25 1/2" because of the overlapping of materials.

Therefore, I suggest that you copy to a new drawing, the box that you drew to represent the cabinet in your kiosk. (You can use the copy command or the wblock.) Then assemble the panels on another layer inside that box. Use the box as an outline for size and freeze or remove the layer when the construction is completed or you will have trouble with the solview commands.

You begin construction from the bottom up.
1 The base for the cabinet to set on.

1 It needs to be able to be leveled when the floor is not.
2 It needs to have some toe space to protect the cabinet. (see mall specs.) The cleaning equipment should not bang up the base not the cabinet.
3 It needs to withstand the liquid cleaners that will be used on the floor.

The base is essentially independent from the cabinetry itself - it's the foundation the cabinet maker will build on.

An example to get you started:

a Begin the base by constructing a 3D box - 3 1/2" x 3/4" x (The length of the cabinet)
b Construct the stretchers using 3D box - 3 1/2" x 3/4" x (the depth of the cabinet, minus 3 1/2" for the kick space (7" if you have kick space on both sides of the counter), minus the width of the side pieces (3/4" each - because the stretchers need to fit inside the base)
If your cabinet is 24" deep and you have 2 kick spaces, the stretchers need to be 24" - 7" - 1 1/2" = 16"

2 The body of the cabinet

This would typically include - Doors
Shelves
Drawers

Doors

Simple doors can be constructed of 3/4" stock (Cabinet door face)
You represent them using the box command.

You need to decide if they are mounted to the front of the cabinet or flush with the front of the cabinet.
Save this as a block.

Shelves
For the shelves, you want to refer to the table above to find the appropriate size.
Example: Over 3'-6” in length and adjust = 1” thickness
You represent them using the box command.

Save this as a block.

Drawers
This is going to take more drawing. Let's say, we need a drawer -- 24” wide, 22” deep and 4” high.

Look at the table above for Drawers....
Side, sub fronts, and backs - premium - 1/2”
Bottoms - over 18” - 1/4”
Front - would be the same as the doors 3/4”

a Construct the front of the drawer using the box command 24” x 3/4” x 4”
b Construct the rest of the drawer by:
1 Construct a box 24” x 22” x 4”
2 Construct a second box that will be subtracted from the first box - this box will have to be the size of the space in the drawer. To calculate this subtract the sides, subfront, back, and bottom of the drawer. The box size is:
- 24” - 1/2” - 1/2” = 23”
- 22” - 1/2” - 1/2” = 21”
- 4” - 1/4” = 3 3/4”

Now locate this second inside the first box - centered from front to back and side to side and 1/4” from the bottom.

Finish the drawer with the subtract command - first selecting the large box and enter and then select the smaller box and enter.

Save this as a block.

The top of the base cabinet:
The choice of the top and how it's attached to the body is critical
1 Consider the problem of chipping
2 Consider the problem of liquid spills
FOR SUGGESTED MINIMUM SIZES REFER TO THE FOLLOWING TABLE

a. Check the table - (Tops and Bottoms - premium) for the wood size - 3/4”
b. The employee side and the customer side need an edge - usually 1 1/2”
c. Construct the top with the box command - 4’ x 2’ x 3/4”
   (notice this will make the top flush with the cabinet below)
d. Now for the edge, use the box command - 4’ x 2” x 3/4” (you will need 2 of them)
e. Block the top with its 2 edges

Table 2: Suggested minimum sizes for parts of a cabinet

<table>
<thead>
<tr>
<th>Frame Cabinets</th>
<th>Thickness</th>
<th>Frameless Cabinets</th>
<th>Thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td>Face frame</td>
<td>3/4”</td>
<td>Face Frame</td>
<td>3/4”</td>
</tr>
<tr>
<td>End and divisions</td>
<td></td>
<td>End / Divisions</td>
<td>3/4”</td>
</tr>
<tr>
<td>Flush overlay</td>
<td>3/4”</td>
<td>Economy</td>
<td>1/2” min</td>
</tr>
<tr>
<td>Economy</td>
<td>1/2”</td>
<td>Custom Premium</td>
<td>5/8” min</td>
</tr>
<tr>
<td>Custom Premium</td>
<td>5/8”</td>
<td>Paneled Construction</td>
<td></td>
</tr>
<tr>
<td>Shelves</td>
<td></td>
<td>Styles &amp; Rails</td>
<td>3/4”</td>
</tr>
<tr>
<td>Economy</td>
<td></td>
<td>Panel</td>
<td>1/4”</td>
</tr>
<tr>
<td>Solid stock or particaleboard</td>
<td>3/4”</td>
<td>Shelves</td>
<td>3/4”</td>
</tr>
<tr>
<td>Plywood</td>
<td>5/8”</td>
<td>Veneer Core Plywood</td>
<td>5/8”</td>
</tr>
<tr>
<td>Custom and premium</td>
<td>3/4”</td>
<td>Center support for shelves</td>
<td>3/4”</td>
</tr>
<tr>
<td>Length</td>
<td></td>
<td>Maximum span</td>
<td>4’-0”</td>
</tr>
<tr>
<td>Over 3'-6” in length and adjust</td>
<td>1”</td>
<td>Particleboard core</td>
<td>3’-6”</td>
</tr>
<tr>
<td>Over 4'-0” in length and adjust</td>
<td>1”</td>
<td>Over 3’-6” use</td>
<td>1”</td>
</tr>
<tr>
<td>Tops and Bottoms</td>
<td></td>
<td>Tom / Bottoms</td>
<td>3/4”</td>
</tr>
<tr>
<td>Economy</td>
<td></td>
<td>WEB Frame / Stretcher</td>
<td>3/4” x 2”</td>
</tr>
<tr>
<td>Custom and primium</td>
<td>3/4”</td>
<td>Particle or Medium Density Fiber</td>
<td>3/4” x 3”</td>
</tr>
<tr>
<td>Length over 4’-0”</td>
<td>1”</td>
<td>Security &amp; Dust Panels</td>
<td>1/4”</td>
</tr>
<tr>
<td>WEB members or stretchers</td>
<td>3/4” x 2”</td>
<td>Medium Density Fiberboard</td>
<td>3/4”</td>
</tr>
<tr>
<td>Backs</td>
<td></td>
<td>Backs</td>
<td></td>
</tr>
<tr>
<td>Economy untempered hardboard</td>
<td>1/8”</td>
<td>Economy</td>
<td>1/8”</td>
</tr>
<tr>
<td>Custom and primium - Plywood or tempered hardboard</td>
<td>1/4”</td>
<td>Custom / Primium Exposed Backs</td>
<td>1/4” 1/2”</td>
</tr>
<tr>
<td>Exposed Backs</td>
<td>3/4”</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Breadboards</td>
<td>3/4”</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Drawers</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Side, sub fronts and backs</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Economy</td>
<td>7/16”</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
DETAILING

At this point we want to consider the cabinet’s detail. A detail for this project means something that makes the cabinet unique.

Here is where the uniqueness and innovation is likely to be found:

a  Innovative hardware and/or use of hardware
b  A unique motif like a design inlayed into the top
c  An unusual joining technique - dovetails, box joints, miters, etc.
d  A unique door and/or drawer front
e  A unique combination of materials - the joining of wood and steel for example
   (remember - it has to be visible in the drawing)

The detail could be any one of a number of other things you may have done to distinguish your cabinetry.

To describe any 3d object in a way that can be scaled for construction, one needs to provide a plan view and an elevation and perhaps a side elevation - - the detail is not presented as only an isometric view.

BEGIN A STUDY OF CABINET DETAIL

1  You are going to have to start looking at other designers’ work. You may find a unique detail as you are cabinet shopping
2.  But it is more likely that it will take more searching, since all creative solutions are the result of a new association. They result from putting two things together that were not together before. To accomplish this, you want to look outside of the obvious. Look for hardware that is used on boats, or appliances, or mechanics tool storage, etc. You will need to look outside the interior design profession.
3  Always try to look at things from an opposite point of view. If the hardware projects out, try having it recessed - - THINK IN OPPOSITES.
4  Look at the surface treatments. Look at the “wrapped” cars that have taken billboard advertising and put it on the surface of the car - - people are looking because it works.
5  Look at the assembly techniques (joints) Quick release - bonded - etc. etc.

Table 2: Suggested minimum sizes for parts of a cabinet

<table>
<thead>
<tr>
<th>Frame Cabinets</th>
<th>Thickness</th>
<th>Frameless Cabinets</th>
<th>Thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td>Custom and primium</td>
<td>1/2”</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bottoms</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Economy (18” maximum width)</td>
<td>1/8”</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Economy (over 18”)</td>
<td>1/4”</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Custom and primium</td>
<td>1/4”</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cabinet door faces</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>All grades</td>
<td>3/4”</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>


Check out the website: http://www.awinet.org/
   This is the Architectural Woodwork Institute.
   Do a search on Woodwork Institute for more websites.
HOW MUCH OF THE DETAIL DO I NEED TO DRAW?

You will want to include the following:

a. A drawing that shows the unique detail.
   This drawing needs to include:
   1. Materials
   2. Hardware
      Hinges
      Drawer slides
      Door slides
      Door and drawer pulls
      Locks
   3. Joining - construction method
b. Notes that point out the uniqueness of the detail

Focus your time and energy on modifying your preliminary cabinet to include the necessary details. Figure out for yourself exactly how much and what you need to draw to get your idea across.

HOW TO MODIFY THE CABINET TO INCLUDE YOUR DETAIL

Now you MAY need to go back and modify your cabinet drawing.

a. The joining to the subfront, back, sides and bottom of the drawers will need to be constructed to:
   1. accommodate the drawer slides
   2. show the preferred joining technique (i.e. butt joint, miter, etc. - see Ching chapter 10)

b. The front of the drawer may need to be modified to include new detail - is the front flush with the cabinet or is it protruding?

c. Are there drawer pulls?

d. The doors need to be modified to accept hinges, latches, handles and locks

e. Tops may need inlays or splash boards

Construct the detail inside the block as you constructed the panels inside the box.
If you have several drawers, only ONE needs to be completely detailed.
If you have several doors with specific detail - framing or inlay - only ONE needs to be completely detailed.

Go to AutoCAD.

Open <-
Find and select the file cabinet_template.dwg

Saveas <-
Save this file as cabmodel.dwg to your personal AFS site (P:drive) or floppy disk (A:drive).
From this point forward, continue working on this new cabmodel file. Keep your old cabinet drawing file as backup.
Tilemode to model space and insert your cabinet model on layer 0.

**Tilemode**

- New value for TILEMODE <0>: 1

**Layer**

- Confirm that layer 0 is the current layer

**Insert**

- Do not use XREF - the solid presentation process will not work with an XREF!

  From the Insert box, push File...
  Find and select your cabinet model
  Push Open
  Push OK Insertion point: 0,0
  X scale factor <1> / Corner / XYZ: 
  Y scale factor (default =X): 
  Rotation angle <0>:

  Explode your model so you can access the solid shapes. Do NOT explode more than one time or your solid objects will revert to regions or planar surfaces. If you explode one more time, these surfaces will turn into separate lines and all 3D solid properties of your cabinet will be lost.

  Be careful when exploding!

**Explode**

- Select objects: pick your model
- Select objects:

**Qsave**

- INCLUDE 2 VIEWS, ONE WITH THE HIDE COMMAND AND ONE WITHOUT

  include any necessary notes on your plot. These may include a statement of your detail concept, materials, typical jointing, special hardware needs, how a detail relates to the overall concept, etc.
Task [13]

Cabinet elevations & sections
The task this week

Construct the plan, elevations, and sections using AutoCAD Solview and Soldraw commands to generate the drawings

To complete this task you will need to

Listen to the instruction on the movie
Take time to further develop and revise the directive and concept statements
Place the completed work in the appropriate locations

The tools

Movies
TA support in labs and on line
SOLVIEW is a powerful AutoCAD command that creates floating viewports using orthographic projection to lay out multi-view and sectional view drawings of 3D solid objects while in a paper space layout.

SOLVIEW calculates the projection as it guides you through the process of creating orthographic, auxiliary, and sectional views.

Information specific to a view is saved with each viewport you create.

SOLVIEW creates layers that SOLDRAW then uses in order to place the visible lines and hidden lines in each view.

**WARNING:** The information stored on these layers is deleted and updated when you run SOLDRAW.

SOLDRAW generates profiles and sections in viewports created with SOLVIEW.

Silhouettes and edges are generated for all solids and portions of solids behind the cutting plane. For sectional views, cross hatching is created.

**WARNING:** Any existing profiles and sections in the selected viewport are deleted and new ones are generated every time SOLDRAW is issued.

All layers, except those required to display the profile of section, are automatically frozen by AutoCAD in the appropriate viewport.

**Steps for Executing the SOLVIEW & SOLDRAW Commands**

Once your cabinet has been designed in model space with 3D SOLID objects, let AutoCAD display your cabinet with hidden lines and hatching calculated for you.

```
Open <- Find and select your cabinet drawing
This is the cabinet model that you attached to the template in an earlier task.
```

Tilemode to model space and load the linetype HIDDEN2.

```
Tilemode <-
New value for TILEMODE <0>: 1 <-
```

```
DDtype <-
Push Load...
Find and select HIDDEN2
Push OK
Push OK to exit the Layer and Linetypes Properties dialog box
```

Before leaving model space, set the view to FRONT and the UCS parallel to this view.

```
DDvpoint <-
Set the X Axis to 270 and the XY Plane to 0
or
Use the pull-down menu option: View, 3D Views, Front
```

```
UCS <-
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/World: view <-
```

Tilemode to paper space and set up the drawing environment.

```
Tilemode <-
New value for TILEMODE <1>: 0 <-
```

Confirm that the limits are set to 24 by 36 for you will print this drawing to both a small scan and a large plot. Erase any existing mviews from your paper space presentation.

Current layer: 0
PRESENTING YOUR SOLID CABINET

When you form the "mviews" through the solview command, make them BIG. You will add dimensions later that will be outside of your cabinet drawing and you would want them to appear inside the mview with plenty of space around them.

**Solview**  
Creates floating viewports and projects multi- and sectional-view drawings of 3D solids

- SOLVIEW creates orthographic, auxiliary, and sectional views that are projected in paperspace. If you are not working in paperspace, SOLVIEW will turn tilemode off and enter paper space mode for you.
- The step that asks you to name the view also provides the names for the layers.
- The layers created by SOLVIEW are updated during the SOLDRAW command. Do not put any drawing information on these layers as they are deleted and updated by AutoCAD automatically
- Layers created by SOLVIEW:
  - view name-VISVisible lines
  - view name-HIDHidden lines
  - view name-DIMDimensions
  - view name-HATHatch patterns used in section views
  - VportsMview borders
- 3 options:
  - UCS A profile view is created relative to a user coordinate system
  - Ortho By picking two sides of a viewport, you can “fold” an orthographic view and project the side you prefer
  - Auxiliary Creates an auxiliary view which is projected onto a plane perpendicular to one of the views and inclined in the next view

Create the front elevation first in the center of the template. This example of the endchest will be shown at 3"=1’0". Depending on the size of your own cabinet, the scale used may be different. Pick an appropriate scale that displays your cabinet large enough to see but small enough to fit within the template border. For example, at 1/2” scale, you will scale to 1/24. For 1” scale, you will scale to 1/12. See “scaling in architectural scale” in the book Using AutoCAD in Your Job: Part I.

**FRONT ELEVATION**

Solview <-

Ucs/Ortho/Auxiliary/Section/<eXit>: ucs <-
Named/World/?/<Current>: <-
Enter view scale<1.000000>: 1/4 <- (these files will be presented at 3"=1’0", so this is the equivalent to setting the scale with the zoom command...remember zoom, 1/4xp?)
View center: pick a point above the label FRONT ELEVATION to locate the center of the view
View center: <-
Clip first corner: pick a point to form the lower left corner of the “mview”
Clip other corner: pick a point to form the upper right corner of the “mview”
View name: front <-
Ucs/Ortho/Auxiliary/Section/<eXit>: <-

You are now in paperspace, looking at your front elevation inside an mview.
Note: There is nothing hidden yet - - you are not done!

**PLAN VIEW**

Solview <-

Ucs/Ortho/Auxiliary/Section/<eXit>: ortho <-
Pick side of viewport to project: pick a point on the TOP of the mview border you have just created (the Midpoint OSNAP will automatically be activated to help you pick the midpoint of the mview border)
View center: pick a point above the label PLAN VIEW to locate the center of the view (Ortho will be automatically activated)
View center: <-
Clip first corner: pick a point to form the lower left corner of the “mview”
Clip other corner: pick a point to form the upper right corner of the “mview”
View name: plan <-
You are still in paperspace, looking at your front elevation and plan view inside their own mviews.

RIGHT ELEVATION

Ucs/Ortho/Auxiliary/Section/<eXit>: <-

Pick side of viewport to project: *pick a point on the RIGHT of the mview border of the front elevation* (the Midpoint OSNAP will automatically be activated to help you pick the midpoint of the mview border)

View center: *pick a point to the above the label RIGHT ELEVATION to locate the center of the view* (Ortho will be automatically activated)

Clip first corner: *pick a point to form the lower left corner of the “mview”*

Clip other corner: *pick a point to form the upper right corner of the “mview”*

View name: *right <-

You will be cutting your section parallel to the front elevation, as if the front of your cabinet were to be removed.

SECTION AA

Place your crosshairs inside the plan view mview and construct section AA.

Mspace <-

UCS, world <-

You will be cutting your section parallel to the front elevation, as if the front of your cabinet were to be removed.

*Solveview*<-

Ucs/Ortho/Auxiliary/Section/<eXit>: section <-

*Turn Ortho ON*

Cutting Plane’s 1st point: *pick a point on the left of the PLAN VIEW as the first point of your cutting plane*

Cutting Plane’s 2nd point: *pick a point on the right side of your cabinet* (you should be creating a straight cutting edge because ORTHO is on)

Side to view from: *pick a point BELOW your imaginary cutting plane* (imagine that you will be “removing” the part of your cabinet between the cutting plane you just defined and the point you are selecting)

Enter view scale < .25 >: <-

*Turn Ortho OFF*

View center: *pick a point above the title SECTION AA*

"If you cannot see the area near the bottom edge of the template, use the scroll bar on the side of your screen or issue a transparent PAN to shift the drawing"

Clip first corner: *pick a point to form the lower left corner of the “mview”*

Clip other corner: *pick a point to form the upper right corner of the “mview”*

View name: *secaa <-

You are still in paperspace, looking at your front elevation, plan view, right elevation and section AA inside their own mviews.

The section view will look identical to the front elevation at this time.

Move the Section AA mview to align with the front elevation. Get into pspace and use the move command. Pick on the mview border to move the section.

Go to your layer box and take note of the new layers created. To help you visualize these solid drawings, change each of the
hidden layers to color red and linetype hidden2. The quickest way to do this is with the keyboard command -layer. Use the wildcard character (*) to get ALL of the hidden line layers at once.

-Layer <- (don’t forget to put the dash in front of the word ‘layer’)
  ?/Make/Set/New/ON/OFF/Color/Ltype/Freeze/Thaw/LOck/Unlock: c <-
  Color: red <-
  Layer name(s) for color 1 (red) <0>: *hid <-
  ?/Make/Set/New/ON/OFF/Color/Ltype/Freeze/Thaw/LOck/Unlock: L <-
  Linetype (or ?) <CONTINUOUS>: hidden2 <-
  Layer name(s) for linetype HIDDEN2 <VPORTS>: *hid <-
  ?/Make/Set/New/ON/OFF/Color/Ltype/Freeze/Thaw/LOck/Unlock: <-

Do not alter the linetype scale of the lines or the LTSCALE command.
The hidden lines will not appear until your presentation is complete and plotted to the correct scale.

**LETTING AUTOCAD COMPLETE THE SOLID VIEWS**

**Soldraw**

*Creates profiles and sections in viewports created with SOLVIEW*

- Soldraw will only work in viewports created with the SOLVIEW command.
- Any existing profiles in SOLVIEW viewports are deleted as the new ones are generated.
- This command uses information generated from SOLVIEW.
- SOLDRAW will freeze all layers inside the viewports except those created by SOLVIEW.
- Places the visible lines, hidden lines and section hatching in the appropriate views.

Soldraw <-
  Select viewports to draw:
  Select objects: pick ALL of the mview borders created by SOLVIEW
  Select objects: <-

AutoCAD will select the solids inside each view and construct the hidden lines.

There is an additional solid command called Solprof.
We will not have the opportunity to use it in this exercise, but be aware of its capabilities.

**Solprof**

*Creates profile images of 3D solids*

- A profile image created by Solprof will display only the EDGES of the solid(s) selected for the current view.
- Solprof will only work with solid objects, not solid surfaces.
- You must be inside an mview in paper space to use Solprof.
- Hidden profile lines can be stored on separate layers:
  PV-viewport handle: Visible profile layer
  PH-viewport handle: Hidden profile layer
- Profile lines can be projected onto a plane as 2D objects (Y) or 3D objects (N).
- A tangential edge is the line that forms as a transition from one tangent face to another (i.e. the lines that appear when you have a curving surface or face). Selecting Y at this prompt will eliminate these lines and give a more full, realistic appearance.

**COMPLETING THE PRESENTATION**
Layer <-
Create a new layer called *pborder* with a color of your choice
Set *pborder* as the current layer

Create an mview on layer *pborder* for your isometric view.

*Mview* <-
ON/OFF/Hideplot/Fit/2/3/4/Restore/<First Point>: *pick a point above the label ISOMETRIC VIEW to form the lower left corner of your mview*

Other corner: *pick a point to form the upper right corner of your mview*

*Mspace* <-
Move your crosshairs into this new mview.
Use the pull-down menu option: View, 3D Views, Select either the SE or SW isometric view.

*View* <-
/?/Delete/Restore/Save/Window: *s* <-

View name to save: *iso* <-

Layer <-
Set 0 as the current layer
Freeze the *pborder* layer
Freeze the VPORTS layer (created by AutoCAD during the solview process)

Draw a section line in paper space (use the LINE command) to indicate your cutting plane through the front of your PLAN VIEW. Use Ortho to ensure your line is straight.

Label the ends of the line with an *A* and an *A* in 1/4" text.
CONSTRUCTION DETAILS

You can create various construction details using regular mviews and viewpoints. Or you can use solview to construct closer views of sections and areas with hidden lines. To keep the borders around these views, create a new layer called pborder2 and change the borders to this layer (use PROPERTIES or DDMODIFY). Don’t freeze it!

Add the necessary information into the notes area using the ROMANS style with the simplex font at 3/16” height.

Edit the scales under the drawing labels to coincide with the scales used in the drawings. So if you created the solviews with a scale of 1/12, dedit the “scale” label to 1”=1’0”.

Edit the titleblock:
Drawn by: Your initials
Sheet No.: 1 OF 1
Approved. by: JV
Date: Today’s date
Job No.: 2006-1

APPLY THE SOLVIEW & SOLDRAW COMANDS TO YOUR CABINET

- Make a backup copy of your cabinet drawing (on a separate disk).
- This is in addition to the cabinet drawing on your personal AFS site.

Follow the procedures you just learned and apply them to your cabinet drawings.
Task [14]

Cabinet Dimensions
The task this week
  Fully dimension your cabinet

To complete this task you will need to
  Listen to the instruction on the movie
  Take time to further develop and revise the directive and concept statements
  Place the completed work in the appropriate locations

The tools
  Movies
  TA support in labs and on line
DIMENSIONING BASICS

HERE ARE THE GOALS FOR SUCCESSFUL DIMENSIONING

1. Include ALL the necessary information
2. Make the dimensions easy to find and read
3. Include ONLY the information needed
4. Make the drawing visually dominate the dimensions

HERE IS THE STRATEGY FOR ACHIEVING THESE GOALS

1. Determine the views and scales the plot must include
2. Set up a layer for each view or plane you will dimension
3. Set up a text style for your dimensions
4. Set up the dimension style
5. Set up and save a view for each anticipated paper space mview
6. Set up the mviews in paperspace
7. Dimension each of the views

SET UP AUTOCAD TO DIMENSION

Open your cabinet file. This is a continuation of your solid cabinet with solviews.

1. Determine the views and scales the plot must include
   Begin with the template representing your plot.
   Sketch out to scale, locate, and label each of the views you anticipate.
   (You need a clear view of the anticipated outcome and how it fits on the sheet).

2. Set up a layer for each view plane you will dimension
   If you created your views with the solview process, you have dimension layers created all ready.
   Proceed to step 3.
   If you have no layers created yet, continue with step 2.
   Enter the Layer dialog box and create the following new layers with colors of your choice:
   - dim_pln Plan view dimensions
   - dim_d_pln Detail plan view dimensions
   - dim_f_n Front elevation dimensions

3. Set up a text style for your dimensions
Text style settings will OVERRIDE any dimension settings. It is best to leave your text style height at 0. If you lock in the height of your text during the style command (i.e. 1/8"), your dimension text height setting will be ignored (i.e. it will not recognize 1/16" as your dimension height).

Your menu should look like this one:

4. Set up the dimension style
DDim

Dimensions dialog box for creating dimension styles

• Dimension styles control the appearance of your dimensions.
• Dimension styles can be created, saved and restored for use later. All setting will be retained when saved.
• The subcommand DIM will allow you to access all dimension settings from the keyboard and from past releases of AutoCAD. To enter the subcommand, type dims on the command line and press enter. To exit the subcommand, type exit or press the Esc key.
• The DIM subcommand will allow you to construct dimensions and change the text during the command. Without using this subcommand, the computer will calculate the dimension length and place it in the dimension automatically.

Follow the instructions for changing the dimension defaults to reflect architectural dimensioning.

Use the pul down menu to find - Format - Dimension Style . . .

The following screen will appear:

Dimension Style Manager screen -

Notice 2 things:
- The default defined style STANDARD is displayed
- A dynamic Preview of dimensions as specified appears

Select New

Enter a name for the dimension style you are about to specify. For this project, call it CABINET.
The style name CABINET will then recall all the specifications that follow.
This will be used for ALL dimensions unless you use the flyout menu (Use for:) and create unique specifications for one of the sub-options.
When you choose to continue, the following menu with tabs will appear. Set each of the menus as follows.

Use the AutoCAD help button to view the definition for each of the options.

Make changes to this **Lines and Arrows** screen as follows:

- **Dimension Lines**
  - Baseline spacing: 1/8”
  - Extension Lines
    - Extend beyond dim lines: 3/16”
    - Offset from Origin: 1/16”

- **Arrowheads**
  - 1st: Oblique
  - 2nd: Oblique
  - Leader: Oblique

- **Arrow size**: 1/8”

- **Center Marks for Circles**
  - Size: 3/32”

Make changes to this **Text** screen as follows:

- **Text color**: By Layer

- **Text height**: 1/16”

- **Text placement**:
  - Vertical: Above
  - Horizontal: Centered

- **Text alignment**: Aligned with dimension line
Make changes to this **Fit** screen as follows:

**Fit Options**
- **Text Placement**
- **Scale for Dimension Features**  
  Change To  
  Scale Dimensions to layout in paperspace

**Fine Tuning**

Make changes to this **Primary Units** screen as follows:

**Linear Dimensions**
- **Unit format:** Architectural
- **Precision** 1/16"
- **Measurement Scale**
- **Zero Suppression**
- **Angular Dimensions**
- **Zero Suppression**

Make NO changes to this **Alternate Units** screen as follows:
Make NO changes to this Tolerances screen as follows:

At this point you select OK.
This returns you to the Dimension Style Manager screen.
Note that the screen now includes CABINET as a Style and that it is the Current Dimstyle.

The screen appears as follows:
5. Set up and save a view for each anticipated paperspace mview

If you created your views with the solview process, you have dimension layers created all ready. Proceed to step 7.

Create the views needed for your dimensioning presentation. For example, if you will be creating a plan view, an elevation and a detail view, change the viewpoint to the necessary viewpoint and then issue the view command. Use the save option to save each view. Give them logical names like F_elev and Detail.

For example, to prepare the front elevation view:

View, 3D viewpoint, Front

Freeze all layers except the front elevation layer

View, save, f_elev

Repeat the same procedure for the other views of your presentation.

6. Set up the mviews in paperspace

Attach your drawing to a template with the xref or ddinsert commands. Tilemode to paperspace. Create a pborder layer for your mviews. Construct the mviews following your sketch from step one above.

Setting the scales of your views:

Restore each appropriate view within the appropriate mview. Remember to get inside the mviews with the mspace command. Once inside each mview, use the zoom command with the scale option to set the correct scale to each view.

Current layer: dim_pln
Freeze layer: pborder & vports

7. Dimension each of the views

Before you begin to dimension, I suggest you use the Pulldown menu to locate view - - toolbars and select both the dimension toolbar and the object snap toolbar.

Begin dimensioning with the overall dimensions in the X, Y, and Z directions. Align the UCS with the view (use the VIEW option) before you begin dimensioning. Follow a logical progression from overall dimensions, to dimensioning the components, and finally the details (i.e. frame, doors and then door handles).

Imagine constructing each piece - - exactly what would you need to know in order to make that piece?

Pspace <-

Current OSNAP: endpoint
Zoom closer to the bottom of your plan view.

Current layer: dim_pln
Freeze in current viewport: dim_d_pln & dim_e_n

Mspace <-
**Dimlinear**  
*Creates a straight linear dimension*

- Linear dimensions can be either vertical or horizontal dimensions. They will form straight lines, regardless of whether Ortho is on or off.
- Once the two points have been selected, you need to pick a point on-screen for the dimension line to form.
- It makes a difference which point you pick first. If you pick a point on the left and continue by picking a point on the right, the dimension will form in-between these 2 points. But if the dimension numbers are long and do not fit inside the space provided, they will form outside the dimension line. In this example, the dimension line would form and the dimension text would appear to the right, close to the second point selected.
- Dimension text can be rotated to appear parallel to the dimension line.
- The keyboard shortcut is DIMLIN.

Create a linear dimension along the same side of the cabinet, but only measuring the depth of the left side.

**Dimlinear** <-
- First extension line origin or press ENTER to select: *pick the far left endpoint*
- Second extension line origin: *pick the next endpoint*
- Dimension line location (Mtext/Text/Angle/Horizontal/Vertical/Rotated): *pick a point away from the drawing*
- Dimension text = #'-#" (the computer tells you the dimension and will place it on the dimension line)

**Dimbaseline**  
*Creates a baseline dimension from a previous linear dimension*

- A baseline dimension is one that follows a linear dimension. It assumes that the first point is retained for the baseline dimension; therefore, the computer only requests that you select a second point. The request for a second point will continue until you press the Enter key.
- The spacing between the linear dimension and the baseline dimension is determined when you set up the dimension style.
- If you wish to continue from a dimension created previously but not immediately before issuing the dimbaseline command, the computer will request for you to pick the dimension to continue from.
- The keyboard shortcut is DIMBASE.

Create a baseline dimension under the previous linear dimension.

**Dimbaseline** <-
- Specify a second extension line origin or (Undo/<Select>): *pick the far right endpoint of the cabinet*
- Dimension text = #'-#" *(the computer tells you the dimension and will place it on the dimension line)*
- Specify a second extension line origin or (Undo/<Select>): *press enter to complete the command*
Create a linear dimension along the side of the cabinet. Dimension only a part of it, like the depth of the material.

**Dimlinear**
- First extension line origin or press ENTER to select: *pick the far left endpoint*
- Second extension line origin: *pick the next endpoint*
- Dimension line location (Mtext/Text/Angle/Horizontal/Vertical/Rotated): *pick a point away from the drawing*
- Dimension text = #'-#''

**Dimcontinue**  
**Creates a continuous dimension**
- A continuous dimension will draw a related dimension to a linear or angular dimension.
- It assumes the second point of the previous dimension is the first point of the new dimension.
- The continuous dimension will align itself with the previous dimension created.
- You can create many continuous dimensions, one right after another. To complete the command, press Enter.
- If a linear or angular dimension was NOT constructed prior to issuing Dimcontinue, the computer will request that you pick a dimension from which to continue.

**Dimangular**  
**Creates a dimension angled between 2 selected points**
- To determine the angle between 2 lines, arcs or circles, pick the objects and then pick the location of the dimension line.
- The dimension line is drawn as an arc and can be located on either side of the points selected. The same angle number will remain constant no matter which point is selected for the dimension line location.
- DIMANG is the keyboard shortcut.
Dimangular <-
Select arc, circle, line, or press ENTER: pick a line
Second line: pick another line
Dimension arc line location (Mtext/Text/Angle): pick a point to place the dimension; you can place it either on either side of the selected objects
Dimension text = #

Pspace <-
Zoom, extents <-

Get inside the elevation mview.
UCS, View <-

Current layer: set the correct elevation layer as current

---

**Dimcenter**

*Creates a mark in the center of arcs and circles*

- You can choose between using a mark, like a little plus sign, or a center line, which is like a target. You can make your selection in the DDim dialog box.
- You cannot draw to this center mark; it is there for display only.

---

Dimcenter <-
Select arc or circle: pick the arc or circle
**Dimradius**  
*Creates a radial dimension for arcs and circles*

- An arrowhead will point toward the edge of the arc or circle. This indicates the limits of the dimension.
- The dimension text can be inside, outside or attached by leader line to the arc or circle. The pick point you select will determine the placement and appearance of the dimension.
- The dimension text will begin with the letter R in front, indicating that the dimension is a radial dimension. This is in contrast to a diameter dimension.
- DIMRAD is the keyboard shortcut.

**Dimdiameter**  
*Creates a diameter dimension for arcs and circles*

- A diameter dimension will measure through a circle or arc and will place an arrowhead at each end of the inside dimension or at the tip of the leader line.
- The dimension text will begin with the symbol in front, indicating that the dimension is a diameter dimension. This is in contrast to a radial dimension.
- The position of your cursor will determine the location and “look” of your diameter dimension.
- Settings within the DDim Format dialog box can be altered to have the dimension text appear within, outside, or connected by leader line to the circle or arc selected.
- DIMDIA is the keyboard shortcut.
**Dimaligned**  
*Creates a dimension aligned between 2 points*

- Dimaligned is useful for dimensioning lines on an angle.
- An aligned dimension can be created by selecting a line, which will align the dimension with the line automatically, or by picking 2 endpoints, which creates a dimension line manually.
- Using the object option does not work on blocks, but will function with lines, arcs, polylines and circles.
- DIMALI is the keyboard shortcut.

---

**Dimordinate**  
*Creates a dimension line that displays the X and Y coordinate*

- An ordinate point dimension measures from the origin or 0,0 point to the point selected.
- If no option is selected (Xdatum or Ydatum), the computer determines if you are selecting an X or Y ordinate and places the leader line accordingly.
- Use OSNAPs for accuracy.
- The second point selected determines the location of the leader line.
- If the ordinate point selected is in the X direction, the X distance will be displayed. If the ordinate point selected is in the Y direction, the Y distance will be displayed.
- Ordinate dimensions are also referred to as datum dimensions.
- 4 options:
  - **Xdatum**: Determines the X distance and places the leader line in the X direction
  - **Ydatum**: Determines the Y distance and places the leader line in the Y direction
  - **Mtext**: Customizes the dimension text as multiple lines
  - **Text**: Customizes the dimension text as a single line

---

### Dimaligned

First extension line origin or press ENTER to select: *use an OSNAP to select a point*

Second extension line origin: *use an OSNAP to select a point*

Dimension line location (Mtext/Text/Angle): *pick a point on the screen*

Dimension text = #

### Dimordinate

Select feature: *with an OSNAP, select a point on your model*

Leader endpoint (Xdatum/Ydatum/Mtext/Text): *use an option or pick a point on screen for the location of the leader line*
**Dimedit**  
*Modifying existing dimensions*

- Edit existing dimensions from the text itself to altering the angle of the extension lines.
- Special characters can be combined with the dimensions, just like with regular text. You’ll find these under the Symbol option of the dialog box.
- The Other... option (under the Symbol option) will allow you to change to any available font and select any character from that font to incorporate into your dimension text.
- 4 options:
  - Home  Returns the text from a justification to its original position
  - New  Type the new text in the dialog box and it will alter the selected dimension
  - Rotate  Alters the rotation of the text from 1 to 360 degrees
  - Oblique  Alters the angle of extension lines

**Dimoverride**  
*Cancels dimension style settings for the selected dimension*

- Overriding a dimension setting means that you can make a change to one or more existing dimensions without affecting the current dimension style. For example, if the text of one dimension needs to be much larger than all others, override the text setting for this one dimension.
- After entering a dimension variable name, the computer will display that variable's current setting. You can then alter it by entering a new value.
- If you've overridden dimension settings on one or more dimensions, you can reverse the override by process. Issue Dimoverride and use the Clear option.
- For a complete listing of the dimension variables, type DIM Once in the Dim subcommand, type status. A listing of the variables will appear in the text window.
- DIMOVER is the keyboard shortcut.
**Dimtedit**  
Alters dimension text on existing dimensions

- Dimtedit will rotate or move the text of an existing dimension.
- You can visually move a dimension to another location by selecting it and then picking a new location on the screen.
- The center location of the text is not changed when altering the angle.
- 4 options:
  - **Left**: The text is left justified on linear, diameter and radial dimensions only
  - **Right**: The text is right justified on linear, diameter and radial dimensions only
  - **Home**: Returns the text from a justification to its original position
  - **Angle**: Alters the angle of the text from 1 to 360 degrees
- **DIMENTED** is the keyboard shortcut.

**Dimstyle**  
Updates dimension style settings for existing dimensions

- The style created through the DDIM command can be modified with Dimstyle.
- Existing dimensions will be updated to the new settings automatically.
- Under the DIM subcommand, the command to use is UPDATE.
- 6 options:
  - **Save**: Create a new dimension style with the current settings
  - **Restore**: Alter the existing dimension settings by combining them with the current dimension style
  - **Status**: Displays all of the dimension variables and their current settings for the current dimension style
  - **Variables**: Displays all of the dimension variables and their current settings for a selected dimension style
  - **Apply**: Updates selected dimensions
  - **?**: List the available dimension styles
- **DIMSTY** is the keyboard shortcut.

**Dimstyle** <-
Dimstyle dimension style: STANDARD
Dimension style overrides:

*various dimension settings are displayed

Dimension Style Edit (Save/Restore/STatus/Variables/Apply/?) <Restore>: _apply
Select objects: all <-
Select objects: <-

**Other commands used with dimensioning:**

**Explode**  
Exploding a dimension will separate the pieces into individual entities; all properties of the dimension are lost

**DDmodify**  
Modify the usual properties plus the text contents, dimension style and even the geometry, format and annotation of the single dimension selected

**DDedit**  
Edit dimension text
ADD DIMENSIONS TO YOUR CABINET

Continue dimensioning the various parts of your drawing. Be careful which layer is current. When you are finished, get back in paper space and zoom extents. Enter each mview and change or confirm that the other dimension layers are frozen in the current viewport.

The goal is to only see the dimensions for the plan in the plan view mview only, to see the dimensions for the front elevation in the front elevation mview only, etc.

You must include AS A MINIMUM the following dimensions:

ISOMETRIC VIEW
No Dimensions

PLAN VIEW
Overall length of the cabinet
Overall width of the cabinet
Overall length of each cabinet component
Overall width of each cabinet component

DETAIL
A minimum of 2 dimensions

SECTION AA
Overall height on one cabinet component (not overall height of cabinet)

FRONT ELEVATION
Overall height of the cabinet

RIGHT ELEVATION
No dimensions required for this assignment
Task [15]

Cabinet Final
The task this week
Attach specifications to your cabinet

To complete this task you will need to
Listen to the instruction on the movie
Take time to further develop and revise the directive and concept statements
Place the completed work in the appropriate locations

The tools
Movies
TA support in labs and on line
HARDWARE SPECIFICATIONS

WHAT NSPECIFICATION NEED TO BE INCLUDED

For each piece of hardware in your cabinet, gather the following information:
- Manufacturer
- Item number
- Number of pieces needed
- Size
- Material/finish
- Cost

Any information that will be needed for ordering purposes must also be included.

AUTOCAD ATTRIBUTE COMMANDS

ATTRIBUTES & SPECIFICATIONS

- AutoCAD enables the user to extract information that can be merged with other software.
- Attributes are text blocks or “tags” that you can attach to any object in your drawing.
- In many professions, these are referred to as specifications.
- These tags provide additional information in a form that can be plotted on the drawing and/or extracted into reports. For example, information may include model number, color, cost, manufacturer, etc.
- These are known as variable attributes because they are not constant (they vary). Variable attributes are attached to blocks for easy manipulation. When the block is inserted into a drawing, attribute values will be requested and displayed with the block.

CREATING THE TEMPLATE FILE

Before attribute information can be taken from the drawing into the report, a template file should be created. This template file tells AutoCAD what to do with the attributes. This template is not constructed in AutoCAD, but in another program available in UNIX, DOS or Apple format. Accuracy in typing this template is essential. C stands for character and N stands for numeral. The numbers that follow designate the length of the data field. Make sure you use the return key (not tab). In addition, there are no extraneous spaces, letters, or returns.

WINDOWS

The instruction that follows was done on a Windows workstation. The constructed using the system accessory “Wordpad”. Open this software (try under Start, Programs, Accessories, Wordpad).

Under File, select Save.
Change to the TEMP directory or the a:drive (your choice).
In the filename box, type specs.txt (this is a text document)
Change the File of Type: to Text Documents (*.txt)
Push Save

Put Caps Lock on. Make sure you are using zero, not capital O!

BL:NAME C008000 <- block name
DESCATT C030000 <- description
MFGATT C015000 <- manufacturer
MODELATT N015000 <- model number (if your hardware models have characters, alter this to C015000)
FINATT C015000 <- finish material
COSTATT N008002 <- cost per item

Under File, select Save.

Either quit the Wordpad software or let it run in the background as you continue with AutoCAD.
ADDING ATTRIBUTES IN AUTOCAD

Open your cabinet drawing.

Now that your template has been made, you can add specifications to the cabinet hardware in your drawing. If you would like to add information to a block, you need to insert the block, explode it, add the attributes and then redefine the block. This will make the attribute request part of the block. When you insert the block again, it will ask you for additional information.

For this exercise, create small blocks that represent the hardware for your cabinet. These blocks can be as simple as a box or a more complex 3D model with detailing, as long as the dimensions are accurate and it roughly resembles the hardware piece.

Once these small hardware pieces are drawn, BLOCK each one. Insert them into their correct locations on your cabinet.

DEFINING ATTRIBUTES

New layer: hardware
Current layer: hardware
Create a text style with the font simplex.

**DDattdef**

*Creates an attribute definition with a dialog box*

- The attribute tag is the name of the attribute value you typed in your template file. This tag must be the same in both the template and in the attribute. The computer will match the attribute value with its proper template file tag.
- AutoCAD attributes have definition modes that can be set or left as the default. These include: Invisible, Preset, Constant and Verify.
- You can set the text style and height to define the way the attribute appears in your drawing.
- A default value can be listed, in case the attribute information is not available when inserting the block.
- An instructional prompt can be created that will be used when the block is inserted. This will help clarify exactly what information is being requested. Keep these prompts less than 24 characters.
- A block can have more than one attribute attached to it.

Zoom, window close to the first piece of hardware <-

Type the generic tags (i.e. mfgatt, modelatt, etc.) - NOT the final information requested!

Create the first specification for the hardware piece. Be careful where you place the specifications so they do not overlap each other.

DDattdef <-

Type DESCATT at the tag: prompt. Go to the next line by using the tab key or picking with your mouse.

Prompt: enter description
Value: unknown
Under Insertion Point, push the Pick Point button. *Pick a point on the hardware*

Justification: left
Text style: simplex
Height: 1/2”

Rotation: 0
Push OK

The letters DESCATT will appear on the hardware.
Create the second specification for the hardware piece.

Type **MFGATT** at the tag: prompt. Go to the next line by using the tab key or picking with your mouse.

**Prompt:** enter manufacturer

**Value:** unknown

Put a mark near Align below previous attribute.

Push OK

The letters MFGATT will appear under DESCATT.

Continue this process, entering all 5 generic attribute tags.
Remember that when you enter numbers (i.e. cost or model), the default attribute value is 0.00.

**ATTACHING ATTRIBUTES TO A BLOCK**

**Explode**

Select objects: pick the hardware piece

Select objects: -

Do NOT explode more than once or you will explode your 3D solid object back to a wireframe object!

Thaw layer 0 if necessary.

**Point**

A drawing reference point

• Points can look similar to grid points, except that they are part of the drawing and will print.
• You can connect to a point with the OSNAP option Node.
• You can create a point by entering its location in absolute, relative and polar coordinates or by picking a point on the screen.
• Points are useful to use as reference points when constructing blocks.
• You can control the size and style of points by manipulating the point style command (DDPTYPE).

**Point**

Point: pick a place at the center of the hardware

A small dot should appear.
**DDptype**  
**Manipulates the point size and style**
- 20 different point styles are available.
- The size of points can be set relative to your screen (size depends on zoom) or in absolute units (recommended).
- When a point size or style has been altered, a regen or regenall command is required to show the alterations.
- DDptype combines the pdsise and pdmode system variables.

\[ DDptype \]  
From the dialog box, select the point style \( \times \)  
Put a marker near **Set size in absolute units**  
Place your cursor in the Point Size box and change the size to 2"  
Push OK

\[ Regen \]  
Notice how the point you just drew on the hardware is now altered.

Current OSNAP: node

**Combine the point, hardware object and attributes by blocking them together.**

\[ Bmake \]  
Block name (or ?): create a new name for the hardware or use the existing name  
Base point: This is the reference point that you will use when you bring the block back into the drawing later.  
A general rule of thumb is to use the center point for curvilinear objects and a corner point for rectilinear objects.  
Push the Select Point button and pick the point (use the node OSNAP).  
Select objects: pick the point, hardware object and the 5 attribute definitions  
Do NOT check the box “Retain Objects”  
Push OK

Insert the hardware object back into the drawing. Pay attention to the prompts! The computer does not always ask for information in the same order that it was originally created.  

\[ DDinsert \]  
Select Block...  
Select the hardware piece you would like  
Push OK  
Push OK  
Insertion point: place the hardware piece in its correct location  
X scale factor \(<1>/Corner/XYZ: \)  
Y scale factor (default=X):  
Rotation angle \(<0>: \)  
Enter ________: ENTER THE CORRECT INFORMATION FOR YOUR HARDWARE AS IT IS REQUESTED
MODIFYING EXISTING ATTRIBUTES

**Attdisp**

*Controls the display of attributes*

- The attribute display can be turned on or off.
- You may want some plots with the attributes displayed and some plots with the attributes not displayed.
- When turned off, any attribute created after this point will not be displayed on the screen or on a plot. To see the new attributes, turn ATTDISP on.

Attdisp <-
  Normal/ON/OFF <Normal>: off <-

Regen <-

Attdisp <-
  Normal/ON/OFF <Normal>: on <-

Regen <-

**DDatte**

*Edits block attribute information*

- The Edit Attribute dialog box will appear from which any of the attribute tags can be changed.
- DDatte will only modify non-constant attribute tags.

**Attedit**

*Edits block attribute values*

- Attedit will allow you to modify the attribute tags, plus other information like value, position, height, angle, style, layer or color.
- Attributes can be modified one at a time or globally (all at one time).
- Attributes do not need to be displayed to be edited globally (Attdisp, off).

NOTE: The DDedit command can be used to edit attribute definitions by altering its tag, prompt, and default values. But since attributes are usually attached to a block, the block must be exploded before the definitions can be changed.
Another way to present your attribute data is in the form of a printout or database. This allows you to view the aggregate data and to manipulate it when combined with another piece of software. You have three options for output: a comma delimited file (CDF), a space delimited file (SDF) or a drawing interchange file (DXF). You have the option of two commands to extract this data: atrtext or ddatttext. Be familiar with both.

**DDatttext**

- Attext stands for ATTribute EXTraction.
- DDatttext provides a way to take the attributes from a drawing and put them in a text report that you can print.
- DDatttext merges the template file you created earlier with the attributes you’ve just added to the drawing.
- 3 forms in which attributes can be extracted:
  - CDF (Comma Delimited Format) - commas separate data fields
  - SDF (Standard Data Format) - standard field length and standard spacing between data fields
  - DXF (Drawing Interchange Format) - exports block reference and attribute information
- Use the extraction form that is suitable for your needs and can merge easily with other software programs, like dBASE, Lotus, Quattro Pro, WordPerfect, etc.
- For the output file, include the path of the file if working on a network computer (i.e. a:\hardware.txt).

**DDatttext** <-

1. Select Comma Delimited File (CDF)
2. Push Template File...
3. Change to the directory where you created your specs.txt file
4. Select your template file from the list (i.e. specs.txt)
5. Output file is the name of your drawing with the extension .txt. (this should appear automatically on the line); push the Output File button and select a location to save this file, such as the A:drive or your AFS site on P:drive
6. Push OK

NOTE: The message "??? records in extract file" will appear on the command line. If not, double check your template file (i.e. specs.txt) for typos and the attributes themselves for mistakes. It may be necessary to create another template file with a different name (i.e. specs2.txt) and then go through ONLY the ddatttext command again.
PRINTING THE ATTRIBUTES IN EXCEL

Windows
Open Microsoft Excel. (Try under Start, Programs, Microsoft Office, Microsoft Excel).
Issue the open command.
Change “Files of type:” to ALL FILES (*.*)
Open your output file (the name of your drawing with the extension .txt that you just created).
Text Import Wizard
- Step 1 of 3: Push the dot by Delimited; push the NEXT button
- Step 2 of 3: Place a check mark by COMMA; change the TEXT QUALIFIER to a single quote (’); push the NEXT button
- Step 3 of 3: Push the FINISH button

Alter column width to accomodate the attributes.
Erase the items in column A (block names); do NOT erase the column. This column will now be for the QUANTITY of each hardware item.

Consolidate the identical hardware specifications into ONE specification. Enter the correct quantity in the first column and alter the cost for the total of all the items (i.e. if you have 2 door pulls at $10 each, your cost would now be $20).

Insert a row above row 1 and label each column in bold (i.e. Quantity, Desc, Mfg, Model, Finish, Cost).
Highlight the entire COST column. Under Format, select Cells...; In the format cells dialog box, on the number tab, select NUMBER; press OK. The costs of your hardware should now have 2 decimal places.

Save this file as a Microsoft Excel Worksheet (*.xls).
Under File, Print.

If the attributes print on more than one page, adjust the size of the font.

The example provided here is in the Greek font and is provided for a visual formatting reference only.

<table>
<thead>
<tr>
<th>QTY</th>
<th>DESC</th>
<th>MFG</th>
<th>MODEL</th>
<th>FINISH</th>
<th>COST</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>big</td>
<td>abitbigan</td>
<td>450064</td>
<td></td>
<td>3.50</td>
</tr>
<tr>
<td>3</td>
<td>big</td>
<td>abitbigan</td>
<td>4442</td>
<td>copperlike</td>
<td>18.00</td>
</tr>
<tr>
<td>24</td>
<td>abitbigan</td>
<td>98883</td>
<td></td>
<td>110.00</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>abitbigan</td>
<td>88</td>
<td></td>
<td>88.50</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>abitbigan</td>
<td>112</td>
<td></td>
<td>20.00</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>abitbigan</td>
<td>114</td>
<td></td>
<td>10.00</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>abitbigan</td>
<td>116</td>
<td></td>
<td>9.50</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>abitbigan</td>
<td>202</td>
<td></td>
<td>7.25</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>abitbigan</td>
<td>102</td>
<td></td>
<td>4.00</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>abitbigan</td>
<td>V372B</td>
<td></td>
<td>25.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>abitbigan</td>
<td>V372B</td>
<td></td>
<td>36.00</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>abitbigan</td>
<td>338B</td>
<td></td>
<td>44.00</td>
<td></td>
</tr>
</tbody>
</table>

MAKE CORRECTIONS THAT ARE NEEDED
This is the time to see how well you have done in relation to the rest of the class.

Remember that all the projects in the class will be changing in these last weeks.
This is no time to allow yourself slack - -.

Go back again and make sure this cabinet drawing is what you want to submit.
See the course website for cabinet comments.
PLOT YOUR CABINET USING 24 X 36 PLOT

Change your AutoCAD Graphics window background from black to white (see the command OPTIONS) and reload your images (see the command IMAGE).

FINAL CABINET -- CHECK LIST

**Section AA** - black line
- Must be drawn to the same scale as the plan
- Must be keyed to the plan (AA)
- Must show correct hatching in red
- Hidden lines must be visible in .pdf
- Must be in red
- Must be dimensioned in blue
- Must be aligned with front elev.

**Plan view** - black line
- Hidden lines
- Must be visible in jpeg
- Must be red
- Dimensions
- Must have multi dimensions
- Must be blue
- Must show detail key
- Must include section key
- Location of cut
- Direction of the view

**Detail** - black line
- Must be shown in both Plan view and Elevation
- NO ISOMETRIC VIEWS
- Must be drawn full scale
- Must be fully dimensioned
- The size of the dimension font must be the same size as the font in plan view
- Must be labeled and keyed

**Front Elevation** - black line
- Must align with plan, section aa, and right elevation
- Hidden lines must be visible in .pdf
- Must be in red
- Must be dimensioned in blue

**Right Elevation** - black line
- Must align with section aa, and right elevation
- Hidden lines must be visible in .pdf
- Must be in red
- Must be dimensioned in blue

**Note section:**
Include in this section, general notes that would apply to the entire cabinet - not just one view - - for example:
- All panels are 3/4" cherry veneered plywood.
- All joints are to be butt joints unless otherwise specified

**Title section:**
Make sure you included the school logo
Include the title of the project
Include your full name
Include the date