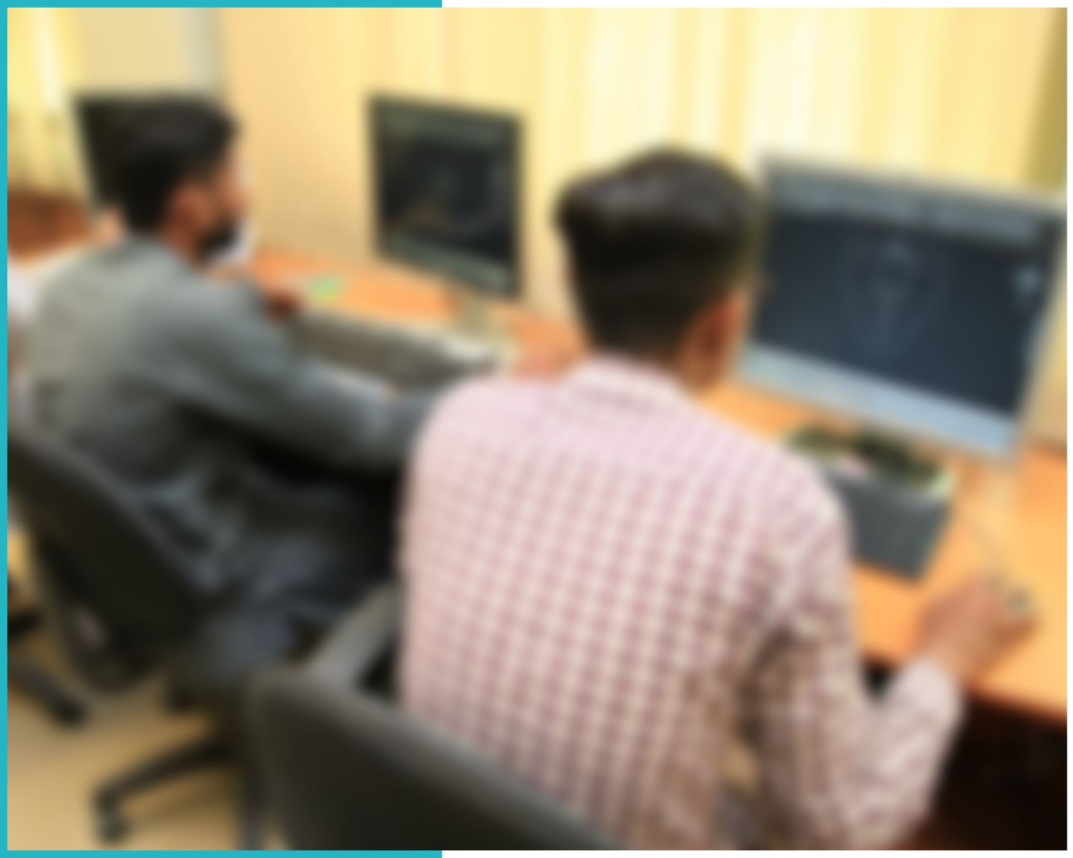
AUTOCAD



LEARNING GUIDE

National Vocational Certificate Level 3

Version 1 - July 2013

EUROPEAN UNION
Kingdom of the Netherlands
german cooperation DEUTSCHE ZUSAMMENARBEIT
NORWEGIAN EMBASSY
Supported by giz Deutsche Gesellschaft für Internationale Zusammenarbeit (GIZ) GmbH
Islamic Republic of Pakistan
Islāmī Jumhūrī-ye Pākistān
NAVTTC

Published by
National Vocational and Technical Training Commission
Government of Pakistan

Headquarter
Plot 38, Kirthar Road, Sector H-9/4, Islamabad, Pakistan
www.navttc.org

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Layout & design
SAP Communications

Photo Credits
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This document has been produced with the technical assistance of the TVET Reform Support Programme, which is funded by the European Union, the Embassy of the Kingdom of the Netherlands, the Federal Republic of Germany and the Royal Norwegian Embassy and has been commissioned by the German Federal Ministry for Economic Cooperation and Development (BMZ). The Deutsche Gesellschaft für Internationale Zusammenarbeit (GIZ) GmbH in close collaboration with the National Vocational and Technical Training Commission (NAVTTC) as well as provincial Technical Education and Vocational Training Authorities (TEVTAs), Punjab Vocational Training Council (PVTC), Qualification Awarding Bodies (QABs)s and private sector organizations.

Document Version
July, 2013
Islamabad, Pakistan

AUTOCAD

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Foreword

The National Vocational & Technical Training Commission (NAVTTC) developed a National Skills Strategy (NSS) after extensive research and consultation with experts and stakeholders including policy makers and representatives from Industry, Academia and the Provincial Government departments dealing with technical and vocational training. The strategy aims at establishing a regime that facilitates competency-based and demand-driven training and assessment.

NAVTTC has developed competency-based training programs with the technical support of TVET Reform Support Programme (TVET-RSP), which is funded by the European Union, the Kingdom of the Netherlands, the Federal Republic of Germany and the Royal Norwegian Embassy. The Programme has been commissioned by the German Federal Ministry for Economic Cooperation and Development (BMZ) and is being implemented by the Deutsche Gesellschaft für Internationale Zusammenarbeit (GIZ) GmbH in close collaboration with NAVTTC. These vocational training programs have been approved by the National Curriculum Review Committee (having representation from all over the country from TEVTAs and industry) for implementation in Public and Private Training institutions.

The purpose of developing competency-based training programs is to equip the learners with modern skills and knowledge for each of the trades to meet the requirements of local as well as international markets. These training programs include competency standards, qualification, curriculum, assessment material and teaching & learning material that will aid in implementation of competency-based and demand driven training in the country.

This Teaching and Learning Material is part of the competency based training program exclusively developed to support the implementation of CBT curricula. This Material is intended for the use of learners undertaking training in *National Vocational Certificate Level-3 in Information Technology (AutoCAD)* under the guidance of experts and appropriate trainers. This Teaching and Learning Material has been designed in consultation with industry, academia and researchers to ensure that the material is relevant and current.

On behalf of the Federal Government/NAVTTC, I wish to express my sincere appreciation and gratitude to all subject matter experts, industry representatives and TVET-RSP experts who have diligently contributed in producing this valuable Teaching and Learning Material.

Executive Director
National Vocational & Technical Training Commission
(NAVTTC)

Introduction

AutoCAD software is used by draftsmen and other professionals for preparing the layouts and designs in architecture industry. It is also used in other industries for designing the layout of the products and their components in three dimensions.

The National Vocational & Technical Training Commission (NAVTTC) has developed a national qualification entitled, “National Vocational Certificate Level-3 in Information Technology (AutoCAD)”. Relevant industry and employers were consulted in the design and validation processes in order to come up with a national qualification that fulfills the requirements of the sector in general and the occupation in particular.

This Teaching and Learning Material (TLM) is developed based on competency standards and curriculum of the AutoCAD national qualification. It carries a learning volume of 80 credits and includes eleven learning modules which are as under:

Module 1: Exhibit Duties & Rights at workplace
Module 2: Perform Orientation about AutoCAD 2D Fundamentals
Module 3: Create 3D Interface Drawings
Module 4: Draw Coordinates
Module 5: Draw 3D orbit Navigation & Model
Module 6: Produce 3D solid & 3D faces
Module 7: Insert Surfaces
Module 8: Develop Solids
Module 9: Modify Solid faces
Module 10: Navigate sections & merge flat objects from 3D models
Module 11: Customize rendering material & light

This TLM provides support for more effective training and productive learning. Each of the learning modules contains learning outcomes and information regarding learning elements in the form of knowledge, skills and attitudes. At the end of every learning module are Frequently Asked Questions (FAQs) and Test Yourself which will help learners in self-assessment before proceeding to modular or final assessment.

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Module 1: Exhibit Duties and Rights at the Workplace

Learning Outcomes

After Completion of this learning module, you will be able to:

- Demonstrate Ethics and Professional Conduct
- Plan Business Process Activities
- Create Awareness about Rights

Learning Unit 1: Demonstrate Ethics and Professional Conduct

Overview

This unit provides the mandatory standard for responsibility, respect, fairness and honesty against the applicable territorial laws and to demonstrates the leaner ethically to understand truth and act in truthful manner in conduct/communication with subordinates in work place.

After completing this unit you will be able to get a complete guide line about the exhibition of ownership in decision making process and the key factors consisting of important steps that will enable you to make different decisions, the practical role of these decisions and observe their results.

Remember

Morality:

The goodness or badness of human acts.

1. Demonstrate Ethics and Professional Conduct

Ethical mindfulness of a worker is a basic element of the work environment for quality of production & services of the business/company.

Ethics at Work Place
Learner should be aware of various basic concepts of ethics relating to their working environment.

Ethics
The word *ethics* comes from Greek word ***ethos*** which mean habits, thus ethics is science of habit of individuals in society.

The proper rules that are implemented on a person's behavior during the work. That demonstrates what is right and what is wrong at workplace, there can be some obligations to stop worker from doing what is not ethical like stealing, not punctual etc.

Respecting the Co-workers, Understanding the Organizational Structure for Departmental, Higher Authorities and lower staff.

Or

Ethics is a sense of freedom with responsibility in behavior.

Or

Ethics is the study of one's acts from the moral perception.

Objective of Ethics:
The objective for studying the ethics is helping the students to have practical knowledge and understanding of self and different ethical issues.

The ethics study establishes the way of one's life/living in a working environment and social environment. That build the worker personal character by learning and adopting good habits for perfection and by learning how to avoid the bad habits that downgrade them.

Naturally Ethic can be divided in to three areas :

1. Normative Ethics
2. Applied Ethics
3. Meta Ethics

Descriptive/Normative Ethics relatively mean that what do the workers think what is right. Normative ethics is how workers should act.

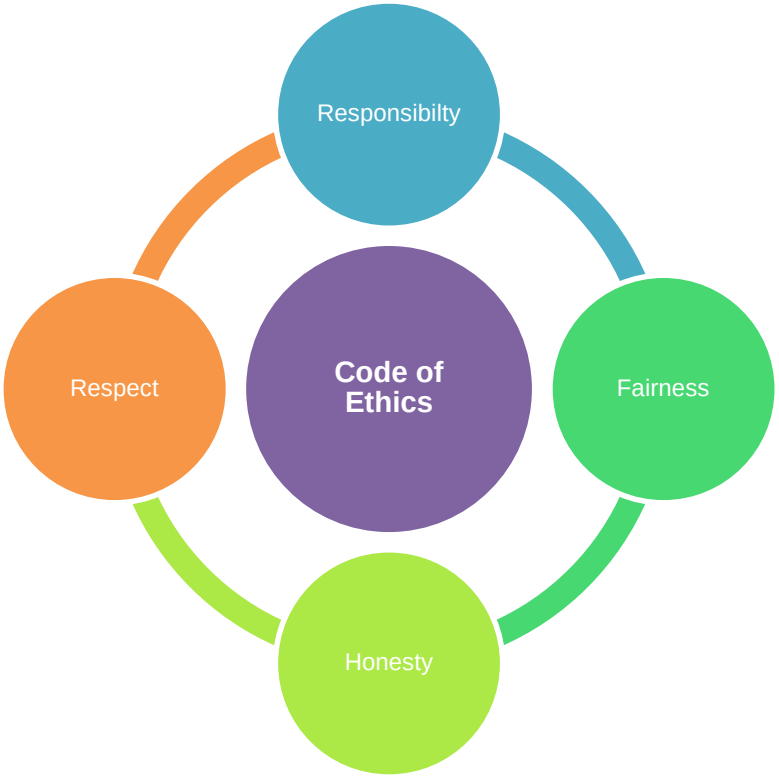
Applied Ethics mean that how do we use moral knowledge and apply it in our practice.

Meta Ethics is about that what actually the right means.

Ethics Training: Training is required to help the workers to understand the ethical aspects of decision making and to include the standard ethics to their daily behavior. It helps employee to deal with ethical issues under pressure.

Code of Ethics:
A formal statement of principles and ethical standards that maintain a worker should Place the integrity of the profession and the interests of clients above his own benefits, Act with responsibility, honesty, proficiency, and respect. Sustain and progress your professional competency.

Remember
Respect
Follow Conduct
Professional Behavior
Safety of work environment
Honesty is the best policy.



Responsibility:

Responsibility is the basic part of job at any level of the employee from top to bottom and bottom to top. Responsibility means taking the owner ship of the decision making or failing to make it, taking owner ship of the actions to take or fail to take action and to take owner ship of the results outcomes of those decisions and actions as well during the work.

The responsibility is to work in best interest of organization, work environment safety and follow the regulations, laws, contracts and sops etc.

Respect:
Respect may be given to worker but due respect that they deserve through their work should be given and one may give respect to the environment working within.

- Learn the working environment, about peoples and their customs and try to avoid the behavior that may be found disrespectful.
- Listen to the workers around you or may contact you in any official regard and try to understand their

point of view or opinion.
- Don't neglect or avoid the workers that you are in conflict or don't agree with, you should contact that worker and resolve the conflicts.

Try to act professionally even when other don't know to act professionally.

Fairness:
Fairness means transparency, equality, impartiality and correct actions in the interest of organization.

- Decision making should be transparent
- Consistence is needed for impartially, objectively take correct actions as and when required.
- Providing equal access to information to workers authorized for having the information.
- Making the opportunities available equally for qualified candidates.
- Hiring , firing , punishing or awarding on must not be made on personal consideration basis
- Avoiding the discrimination of other on the basis of age, gender, language, nationality, color and creed etc.
- Application of rules without favoritism in working environment.

Honesty:
Honesty means that your discussions, communication, commitments and actions are clearly based on truth.

- Your communication and conduct should be based on truthfulness.
- Make your commitments accurately and in timely manner.
- Do not behave like a dishonest and mislead worker.
- Make the working environment to feel safe.

Ethical behavior:
Respect for humanity, respect for basic rights and be good human at work and other place. Excellence and growth based on the talent of the workers.

Professional conduct:
The standards professional conduct are guidelines similar like code of ethics but the standards of professional conduct guide lines are more clear and in detailed way in the professional environment.

Remember

Work gives you meaning & purpose and Life is empty without it.

A professional worker should be bound by certain requirement, certain fundamental issues or principles, rules that govern worker behavior, when we talk about professional that means there is some profession behind it and there is some institution that tell that how to behave like a professional. The professional status comes from the trust that users use in public places in opposition

- A profession gives us a certain set of guide lines to follow
- Professional conduct is the making of rules by professionals of the governing body for worker to follow it.
- The professional conduct is normally taken for the private organization's professionals for contractual nature of employment.
- Professional code of conduct may vary from profession to profession like Teaching, Marketing, Accounts, Medical and Administration etc.

The code of professional conduct assures that a professional has attain and maintain the right level of professional conduct and behavior. So it governs that how you behave, treat people, colleagues, how you deal with things. The trust of others on us to be a professional requires that we must follow and practice the highest standards of the code of professional conduct.

2. Exhibit Ownership of the Decisions

Remember

Implantation of ethics and performing duties are main step of success in any organization

The Exhibition of owner ship in decision making can be explained as the demonstration of their decisions in practical life .It is not necessary that every decision that you will make is successful and it can be failed .We encourage you to make the right decisions and bring the positive prospective in your mind that if “Decision are not failed then their process of learning will stop” and learning process works as a circulation of blood in practical and professional life. Important factor is that you should avoid the negative decision and negative expectation from a positive decision. According to this course **AutoCAD** where requirements and decisions are not pre planned so taking the controllable risk will help you. The main point is to developing a project or working on a project you have to make the decisions by own, you will guide to make the small decisions on your edge that can be easily handled and a forward step towards the successful decisions.

3. High Regard for Resources

Confidence and trust are major skills that a teacher transfer to students and motivate them to work. Assignment of different tasks and goals, Designing and development of different projects and trust them on their resources will encourage you to work towards success. In this professional subject you have to work in different way by using the different equipment provide them a trust shelter so they can easily work without the fear of wrong and faults. Appreciation is also an important factor that is necessary to encourage the team and individual work. Creativity and innovations are very important things in learning process because these bring new results that make the professional or even a student motivates to work further, but it needs trust and confidence that is only provided by teacher.

4. Decisions and Act Impartially/Objectively

Decision making process is no doubt a complicated and tough task but it is a great platform to learn the different activities and projects and develop the different skills .Some people are very active to complete their tasks individually but it is not necessary to work completely on the decisions we can divided them in teams and start working accordingly. There is another factor that everyone not have the similar skills and qualities so while making the teams and start working you will get the pool of skills .When you set a goal, then you will start working on it and this will enable you that how you work partially and impartially work on decisions and Complete them .**Self -assessment or self-examine** is a test regarding your skills in which you have to judge yourself without any interest and assigning projects like **AutoCAD** which you will be cover later in each module .Regardless to professionalism and other formal thinking there is a moral happiness and inner satisfaction that encourage and motivate you to work more and achieve the different tasks.

5. Act in Truthful Manner in Conduct/Communication

Remember

Work gives you meaning & purpose and Life is empty without it.

Communication is a major tool that plays a vital role because through communication you can easily explained your idea your point of view, you’re thinking your knowledge to others. If everyone assume that next person or collogue or co helper or your team coordinators understands your view without any explanations or communication it is very hard to work. Regarding to your course in which team work delegation of different tasks, projects are involved they need proper discussion and communication sessions to share and make the different decisions accordingly. Communication provides everyone a chance to explain their thinking and ideas. Sharing different ideas and discussions are founder of innovations and creativity. Communication works as a stairs to reach the destination.

Learner's Activity

CASE STUDY

Shumaila loves clothes. She wishes to have all the designer clothes. Unfortunately, she can't; she has two brothers and three sisters, and her parents cannot afford to buy all their children designer clothes.

Shumaila has a few friends that have some designer jeans. Her friends let her borrow the jeans, even though this is against her parents' wishes. Her parents felt that Shumaila should not borrow them in case something may be happened to them. In that case, they would need to purchase an outfit to replace the damaged one.

One day on the way home from school, Shumaila tripped and fell on her knees. She tore holes in both knees of the designer jeans she had borrowed from her friend, Julie. Shumaila was crying, not only because she had injured her knees, but because the new jeans were ruined and they weren't even hers.

Write down your opinion about Shumaila decision

1. What is Shumaila going to do?
2. Should she tell her parents?
3. Should she tell Julie?
4. What decisions should Shumaila have made prior to borrowing the jeans?
5. Does Shumaila have an obligation to Julie?

CASE STUDIES WORKSHEET

Use this worksheet to evaluate case study.

1. Identify the problem.

2. What are the person's choices?

   a.

   b.

   c.

   d.

   e.

3. Gather information: What information should the person gather that would be helpful to know before making a decision?

   a.

   b.

   c.

   d.

4. Consider the outcome. What would be the results of the decision?

   a.

b.

c.

d.

5. Make the decision. What should the person do?

Evaluate your decision. Why do you think this is the best decision possible?

6. Ethics in the Workplace

If you are working in an organization, you must consider the following ethical approaches in mind to execute your duty at workplace.

These approaches define the ethics in workplace either you are working in higher management or lower staff.

The following four points help you to understand work ethic in workplace and truth and act in truthful manner in conduct/communication.

1. Relationships at Work
2. Business Abuse
3. Results of Business Abuse
4. How Employers Respond or rectify to Business Abuse

1. Relationships at Work

- Management/owner toward client/customer
- Management/owner toward employees
- Employee toward Employer, Co-workers, Customers

2. Business Abuse

Any illegal, unethical, irresponsible Act done against an Employer

3. Results of Business Abuse

- Show Product prices very high
- Business Failure due to Mismanagement of an organization
- Fewer Jobs
- Unpleasant/Unfriendly Working Conditions and environment of an organization.
- **Examples of Business Abuse**

  - **Example-1**
    - Stealing of Merchandise and Money
    - Shoplift
    - Kickback
    - Conflict of Interest
    - Unauthorized Discounts
    - Time stealing: Come late and leave before timing

  - **Example-2**

    - Destruction of product, record, faith.
    - Falsify Records
    - Break Confidentiality
    - Ignore Safety Rules
    - Misrepresent Merchandise

  - **Example-3**
    - Poor Client Service
    - Irresponsible Behavior

  - Addictions
    - Drugs
    - Alcohol
    - Tobacco

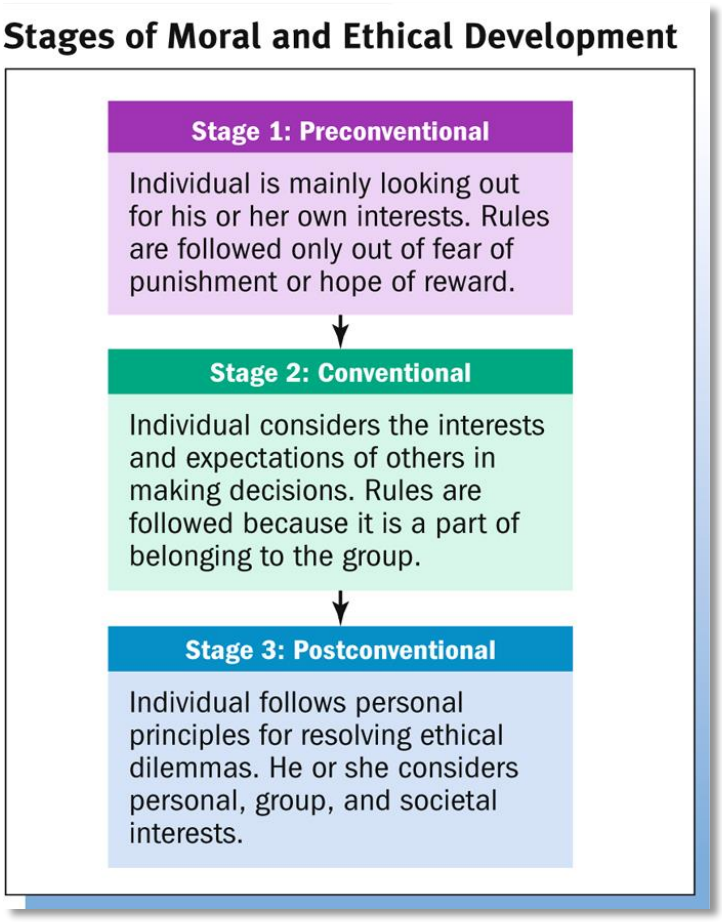
4. How Employers Respond or rectify to Business Abuse

- Increase Security

- Loss prevention management.
- Investigation
- Networks/Helplines
- threats of trial
- **Increase Employee Loyalty**
  - fair remunerations & welfares
  - fair and reliable treatment
  - recognition for good work

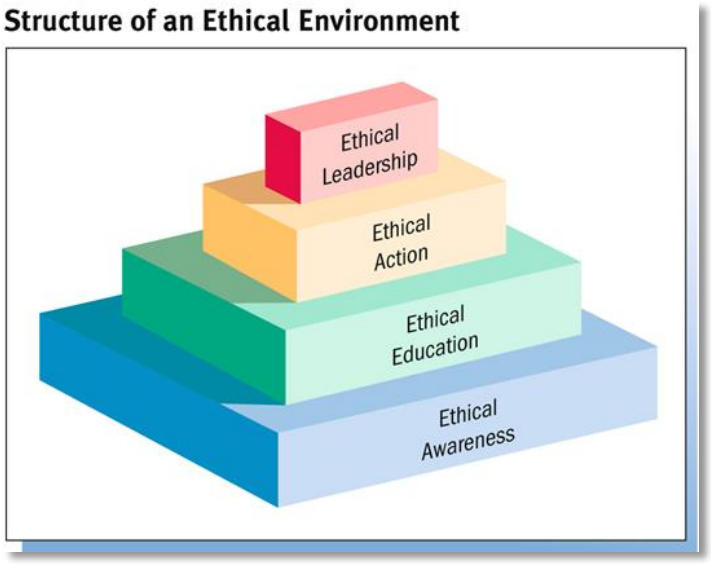
7. Development of Individual Ethics

An individual ethics can be Developed in any environment either you are a student at any institution or employee in any organization; you need to consider the following stages.



On-the-Job Ethical Dilemmas: There are some Ethical Challenges while you are in work place.

How Organizations Shape Ethical Conduct:



Ethical Awareness

Ethical awareness is essential part of an Institution and organization which describes Code of Conduct formal statement that defines how the organization expects and requires employees to resolve ethical questions.

Ethical Education

Codes of conduct cannot detail a solution for every ethical situation, so Institution and corporations provide training and workshop in ethical reasoning as **Character Building society** in Pakistan initiated by Director General (Awareness and Prevention) National Accountability Bureau (NAB).

The Aim of this society is to following a prudent strategy to set up character building societies at grassroots level across the country aimed at making the younger generation useful citizens and enabling them to differentiate between good and bad deeds.

Ethical Action

Ethical education helps the employee's recognized and reason through ethical problems and turning them into ethical actions.

Ethical Leadership

Executives in organization and Trainer in Institution must exhibit/show ethical behavior in their actions while dealing employees and Trainer respectively.

Learner's Activity

Briefly answer the following questions:

1. You have started working for an organization as AutoCAD operator.
    - What might be your duties and rights in accordance with the following conduct requirements?

        - **Integrity**
        - **Objectivity**
        - **Competence**
        - **Fairness**
        - **Confidentiality**
        - **Professionalism**
        - **Diligence**

What impact will policies, rules and regulations governing the work and workplace have on your employment?

Learning Unit 2: Plan Business Process Activities

Overview

This unit provides the complete guide line about the In-line manager role and duties. After completing this unit you will be able to Identify tasks by coordinating recurring meetings, intimate resource availability, create and keep documentations, validate applicable company defined standards.

You will also be able to define tasks and activity to subordinates by applying the complete life cycle methodology and provide time estimation.

1. Communication in the Organization

Communication is used extensively in the managerial functions of planning, organizing, staffing, directing and controlling. Virtually every task that a manager performs requires the use of communication in one form or another.

The nature of communication in the modern organization can be studied by examining the direction of communication flow and the destination of the communication. While in the modern organization, communication flows downward, upward, and the horizontal, its destination can be either internal or external.

Downward Communication:

Activity
Why is Communication Important?

Downward communication flows from people at higher levels to those at lower levels in the organizational hierarchy. The primary function of downward communication is to inform employees about things important to them such as:

Information about their jobs

- Organizational policies and procedures
- Feedback about their performance
- Organizational goals and objectives

The types of downwards communication may include instructions, speeches, meetings, announcements, memos, notifications, letters, hand-books, pamphlets, company newsletters and periodicals, bulletin boards (notice boards) policy statements, and procedures. For downward communication, some use written communication, others use oral communication.

Why is this needed?

When employees receive appropriate downward communication from management, they can be better motivated and become more efficient. While the employees need clear job directions and safety rules, they also need to know facts about organizational strategy, products, and viewpoints on important controversial issues.

Employees want to know about their benefits - health care, insurance, promotions, pensions, training, work environment, retirement, etc. etc. In a way the employees, through their collective pressure, force their employers to be accountable for their decisions through effective downward communication.

Upward Communication:

Upward communication travels from subordinates to superiors and continues up. Upward communication is

extremely important, as upper management needs to know specifically about: production performance, marketing information, and financial data, what lower-level employees are thinking, and so on.

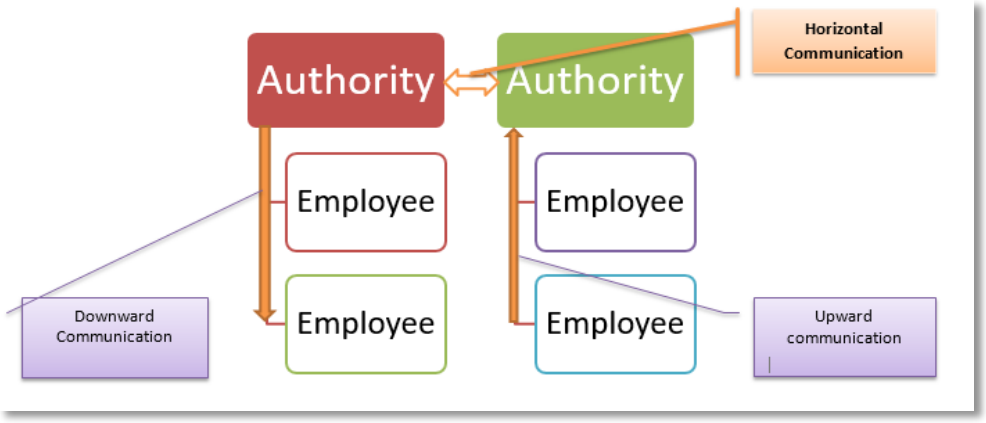
The better the quality of information they receive, the more useful and effective it will be in their decision-making efforts. Unfortunately, this flow is often hindered by people in the communication link who filter the messages and do not transmit all the information, especially unfavorable news to their bosses. Types of media used to direct information upward are reports, interoffice memos, supervisor subordinate conferences, suggestion systems, and grievance procedures.

Upward internal communication is becoming increasingly important day by day. Today many executives sincerely seek frank comments from employees, in addition to the usual periodic reports. Successful managers listen closely to opinions, complaints, problems, and suggestions especially when they are clearly and effectively stated. As a response to increasing global competition, some companies are developing new management styles, which make input from employees an integral part of important decisions affecting the company.

Lateral Communication:

It includes horizontal flow of information, with people on the same or similar organizational levels, and diagonal flow, with people at different levels who have no direct reporting relationships. The lateral communication is used to speed up information flow, to improve understanding, and to coordinate efforts for the achievement of organizational objectives.

Effective lateral communication between peers is essential in organizations to solve problems, perform job duties, prepare for meetings, listening to and making requests, writing notes and memos, and discussing and writing about projects. Most lateral communication is of an oral nature, involving a conference between the participants. In some instances the conference is conducted by telephone.



2. Business Process

A business process is a collection of related activities or tasks that outputs a specific product or services what will be designed for a specific objectives for a particular customers or client.

There are three main types of business processes:

1. **Management processes:** That define the rules regulations of the operation of the system e.g. strategic management.
2. **Operational processes:** That define the basic business for the creation of primary value that is to developing, marketing for the product **or services**
3. **Supporting processes:** That define the recruitment, financing and technical expert.

Activity Definition: A thing that a person or group does or has done. Activity can be specific for specific subjects like if you have to perform an activity in class room, that activity will be related to your class, or you are ask to

differentiate between intra communication and inter communication? That could be related to communication skills.

3. Life Cycle Methodologies

The life cycle methodologies also referred to as systems development life cycle (SDLC), or also called the application development life-cycle, is a term used in systems engineering, information systems, software engineering and project designing to describe a process for planning, creating, testing, and deploying an information system. The systems development life-cycle concept applies to a range of hardware and software configurations, as a system can be composed of hardware only, software only, or to develop a project/system with the combination of both hardware and software.

A life cycle methodologies consists of different work phases which is used by system developers and system engineers to plan for gathering information regarding the system, design , develop , test , implement and generate reports for information.

As we see anything that is to be manufactured on step by step procedure to get the final result, for this purpose life cycle methodologies aims to produce high quality systems that meet customer/client expectations as per customer requirements, by delivering systems which move through each clearly defined phase, within scheduled time-frames and cost estimates.

A number of SDLC prototype or model or methodologies have been created, such as "**waterfall**"; "**spiral**"; "**Agile software development**"; "**rapid prototyping**"; "**incremental**"; and "**synchronize and stabilize**". The oldest of these, and the best known, is the waterfall model: a sequence of stages in which the output of each stage becomes the input for the next. These stages can be characterized and divided up in different ways, including the following:

- **Preliminary analysis**: The objective of phase 1 is to conduct a preliminary analysis, propose alternative solutions, describe costs and benefits and submits a preliminary plan with recommendations.

  Conduct the preliminary analysis: in this step, you need to find out the organization's objectives and the nature and scope of the problem under study. Even if a problem refers only to a small segment of the organization itself then you need to find out what the objectives of the organization itself are. Then you need to see how the problem being studied fits in with them.

- **Systems analysis, requirements definition**: It is the process of gathering and interpreting facts, diagnosing problems and recommending improvements to the system. Analyzes end-user information needs and also removes any inconsistencies and incompleteness in these requirements.

  A series of steps followed by the developer are:
  1. Collection of Facts: End user requirements are obtained through documentation, client interviews, observation and questionnaires.

  2. Scrutiny of the existing system: Identify pros and cons of the current system in-place, so as to carry forward the pros and avoid the cons in the new system.

  3. Analyzing the proposed system: Solutions to the shortcomings in step two are found and any specific user proposals are used to prepare the specifications.

- **Systems design**: Shapes the requirements into a system or design which will done by the system designer and describes desired features and operations in detail, including screen layouts, **business rules**, **process diagrams**, **pseudocode** , system flowchart (a graphical representation of the programs within a system and how they interrelate) and other documentation.

- **Development**: Development can be of different types,

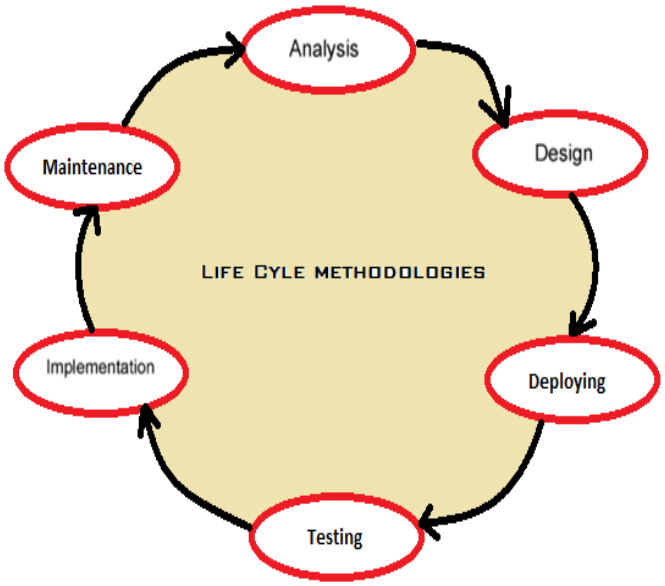
- The real code is written here, if needed.
- Start the real designing, if you have to design something in AutoCAD.

- **Integration and testing**: Test and debug the program/system.
  - Test your work to resolve problems in the system.
  - Unit testing
  - Systems testing or Integration testing

- **Acceptance, installation, deployment**: The final stage of initial development, where the system is put into production and runs actual business.

- **Documentation**: Documentation is extremely important, it should be write an on-going process. After completing the development you should bring the documentation all together.

- **Maintenance**: During the maintenance stage of the SDLC, the system is assessed to ensure it does not become obsolete. This is also where changes are made to initial software. It involves continuous evaluation of the system in terms of its performance.



Learner's Activity

Develop a 2D drawing with Plot area of 30' x 45'

Note: Apply Life Cycle Methodology to develop the above 3D Drawing.

4. Estimated Time

Estimate time is an "estimated time to complete" is a time projection and or effort required to complete a project/task activity in which the task should be completed. Estimated time to complete is a value that is expressed in hours of work required to complete a task or project.

When you are ready to get start with the project you have to plan for the project in initial stages, one of the main objectives is to get a realistic estimation of time to complete. Estimating the time to complete will help you with resource, cost and time planning.

- You can use the work breakdown structure to facilitate estimation.
- You can review the tasks at the bottom of the work breakdown structure and look for the longest path through the sequence diagram.
- You can then use experience from previous projects to estimate the time it will take to complete the tasks.
- If you have the previous data, it will be the key element of precisely estimating time to complete.
- For example, let's assume that you are a CAD Operator and you have to develop a 2D Drawing having plot area 35' X 50'. The drawing will require 5 hours. You review data from previous developed drawing in which the task has taken 5 hours to complete, thus in your drawing you can estimate the time of completion as 5 hours. In the same way you can use past data to estimate the time of completion of an entire project.
- In the absence of previous data or experience you would have to guesstimate, which is inefficient and may error occur in your time guesstimated.
- To develop a schedule, all of the tasks and estimates are placed in a calendar. The calendar would include a specified time period, identify resources and their allocation, how much time the task is expected to take and when each task is scheduled to start and finish as the scheduled shown in table below.

Do You Know!

Breakdown: Break down is used to divide the project into simple modules or tasks to complete the whole project within different dates.

Number	Task Description	Start Date	Completion Date

- The preferred way to complete the project is to apply **"Break down"** structure and **"Time estimation to complete"** on the project.

Procedure
How to apply time estimation and Breakdown Structure?
Develop a 3D drawing with Plot area of 30' x 45'.

Breakdown:

To develop the 3D drawing in 5 days, you have to divide the drawing in different stages.

Stage-1: Development of 2D drawing and coordinate setting
Stage-2: 3D Development
Stage-3: Furniture setting
Stage-4: Rendering and Lighting

Time estimation:

Number	Task Description	Start Date	Completion Date
1	2D drawing and coordinate setting	12/07/2015	13/07/2015
2	3D Development	14/07/2015	14/07/2015
3	Furniture setting	15/07/2015	15/07/2015
4	Rendering and Lighting	16/07/2015	16/07/2015

5. Work Load

Workload is the amount of **work** an individual has to do. There is a distinction between the actual amount of **work** and the individual's perception of the **workload**.

Workload can be of two types one as: **Quantitative:** The amount of **work** to be done as normally you can see in office environment there you work in a specified period i.e. 8 hours in a day. The second as **qualitative**: the work which may difficulty by nature.

Learning Unit 3: Create Awareness about Rights

Overview

This unit provides the complete guide line about the inspirational requirements of human rights in employment context. After completing this unit you will be able to know the rules and regulations under the formation of Standard Operating Procedure (SOP).

Further you will be able to know the different rules, different types of rights and proprietary protection with its different kinds, like copyright and Intellectual rights etc.

1. Rights

Rights are legal, social, or ethical principles of freedom or entitlement; that is, rights are the fundamental normative rules about what is allowed of people or owed to people, according to some legal system, social convention, or ethical theory.

2. Rules

Rules define standards for activities which help the smooth operation of the Project/Task/ or organizational work at workplace.

Definition: Rules is a Statement that establishes a **principle** or **standard**, and **serves** as a **norm** for guiding or mandating **action** or **conduct**.

Rules may be divided into four **general** categories:

1. **Folklore:** These rules are Unpublished that are conveyed by **behavior** and are implicitly understood.
2. **Guidelines**: These rules are commonly **published** and recommended **practices** that allow some **discretion** with their interpretation and use.
3. **Mandates**: These rules are Published commands that may not be ignored in any circumstance and whose **violation** is punished.
4. **Policies:** Published rules that imply a predicted behavior and whose violation may be **permitted** or tolerated under certain circumstances.

3. Responsibility

Definition: A moral, legal, or mental accountability. A particular burden of obligation. Answerable.

Aspects of Responsibility

- Accountability: We are free to choose and that makes us accountable for our choices.
- Duty: We are obliged to do what is expected of us by virtue of our jobs and our work force positions.

Employee's responsibility: The following responsibility must be performed by the employee while working in work place, all these responsibilities will be mentioned in the SOP of the organization, the employee first read the SOP and then take care of each point in the workplace.

Employee should

- Be punctual in his/her duty and honor to his/her commitments for the duty time
- Do what the employer say they will do

- Always listen to what the customer/client wants and treat them politely.
- Honor service commitments
- Treat proprietary information appropriately
- Provide value for the paycheck
- Always honor the agreements with the employers either its formal job agreement or what they have implicitly agreed to do.
- Behave loyally to the employers.
- Take care of proprietary information of the organization.
- Provide good customer/client service.
- Be good agent of the company's resources.
- Commit to organizational goals and try you best to achieve those goals.

Employer responsibility: Following are the responsibilities of the employer toward employees in workplace.

Employer should

- Keep business sustainable.
- Act in accordance with laws/rules/regulations.
- Always provide competitive wages and benefits to employee.
- Continually enhance employee skills which will benefit for both employee and employer.
- Provide reasonable health benefits (e.g., sick, casual, Earn, Leave encashment (If Possible) and maternity/paternity leaves).
- Respect the employees and take care of their dignity.
- Pay a living wage
- Honor contracts and obligations to employees.
- Reinforce and reward organizational loyalty.

4. Standard Operating Procedure (SOP)

A Standard Operating Procedure (SOP) is a document consisting of step-by-step information on how to execute a task.

Standard Operating Procedure (SOP) is a document to define rules and regulations for specific task / project or company profile.

Example: Here we have an example of TEVTA short courses, SOP has recently been developed in all TEVTA Institutions.

Notification released before running the project/task by the board of management, they define the rules and regulation for running of the project.

GOVERNMENT OF THE PUNJAB
TECHNICAL EDUCATION & VOCATIONAL TRAINING AUTHORITY
96-H, GULBERG II,LAHORE PHONE: 042-99263055-59
www.tevta.gop.pk
(OPERATIONS WING)

NOTIFICATION

Subject: - **SOP FOR 3 & 6 MONTHS SHORT COURSES (MAY, 2015) PHASE - II**

No. TEVTA / GM (O-II) / Short Courses / 244 – 15/ 529 The standard Operating Procedure (SOP) is hereby notified for conduct of 3 & 6 months duration short courses (Phase – II) in selected TEVTA institutes. The SOP is attached for implementation in letter and spirit.

No. Even
Dated: May 19, 2015

CHAIRPERSON TEVTA

A copy is forwarded to:

1. All General Managers, TEVTA.
2. All Deputy General Managers, TEVTA.
3. All Zonal Managers, TEVTA, Punjab.
4. All District Managers, TEVTA, Punjab with the request to forward the same to the field formation immediately for further necessary action.
5. The Manager (MIS), TEVTA.
6. SA to the Chairperson TEVTA.
7. PS to C.O.O TEVTA.

1. INTRODUCTION

In line with the policy directions of the Government of the Punjab, TEVTA has specifically focused on requirements of the industry and has started an aggressive, forward looking and demand driven courses to provide skills for employability to the youth of Punjab with focus on children of poor and down trodden families' w.e.f. May, 2015. The short courses have been designed after interaction with the industry / prospective employers.

2. SALIENT FEATURES OF PROGRAMME

To attract the poor unemployed youth for training in employable skills, the program has following features:

- No fee will be charged
- Registration fee will be provided by TEVTA.
- Free training material will be provided by the institutions.
- Stipend will be awarded for financial support of students, on completion of course.

3. TRAINING SCHEDULE FOR 1st BATCH (PHASE – II)

Start of classes of 1st Batch (Phase – II) w.e.f May 18th, 2015.

4. TRAINING PARAMETERS

- Courses duration — 3 months & 6 months
- Total Contact hours — 400 hours / 800 hours
- Average daily conduct hours — 5 hours / day (5 days / week)

4.1 Timings

The training / classes will be conducted in evening shift from 2:00 p.m. to 07:00 p.m. (5 hours per day). However, respective District Manager may accord permission to the institutes' for early start of classes 01 hour, considering the locality of the institute and weather conditions. All the classes will be arranged in the premises of institutes and will not start before 02:00 p.m.

4.2 Number of Trainees per Class

Admission target for each institution has been given by the concerned District Managers, keeping in view demand of the industry in the district. The admission target finalized by the HQ has been circulated among the field offices for execution of training activities, as per plan.

Payment Criteria Show the Payment Method which Never be Changed.

9. ENGAGING / HIRING OF STAFF

9.1 PAYMENT CRITERIA OF TEACHING STAFF

Teaching staff will be paid on basis of per hour. If the enrollment in the class is less than 60% of the capacity mentioned above, no remuneration will be paid to the instructors In case the enrollment is 60 to 80 % of the capacity proportionate payment will be made to the instructors. In case enrollment is above 80% full remuneration as mentioned in para 8.1 will be paid to the instructors.

9.1.1 Admin staff will be paid on the basis of per trainee per month. Expenditure for remuneration to the staff must remain within maximum limits.

9.2 RATE OF REMUNERATION FOR TEACHING STAFF

The engaged teaching staff must be competent and skilled to impart the relevant skills and theoretical instructions.

Hiring of highly skilled persons is recommended from market after a healthy competition if the instructor from institute is not available or not competent to impart training.

RATE OF REMUNERATION FOR ADMIN STAFF

SR. NO.	CATEGORY	MAXIMUM MONTHLY LIMIT
1	Principal / Manager / Head of Institution	Rs. 3000/-
2	*Admission Clerk / Accountant*	Rs. 2000/-
3	Naib Qasid / Attendant	Rs. 1500/-

5. Proprietary Information

Proprietary information is only information which is disclosed in front of the owner only. These information in the form of documents limits who can view it or know about its contents, it also called trade secret.

Some examples of proprietary information include:

- Financial Information
- Employee Document and Profile information
- New Business Plan information
- Stock and Inventory Information

Intellectual property (**IP**) is a term referring to creations of the **intellect** for which a **monopoly** is assigned to designated owners by **law**.

Intellectual property rights include

- **Patents:** A **patent** is a form of right granted by the government to an inventor, giving the owner the right to exclude others from making, using, selling, offering to sell, and importing an **invention** for a limited period of time, in exchange for the public disclosure of the invention.

- **Copyright**: A Copyright is an exclusive rights to a producer/creator/designer of their original works, usually for a limited time, but can be renew by renewal law.

- **Industrial design rights:** An **industrial design right** also called **design right,** protects the visual design of objects that are not purely useful. An industrial design consists of the creation of a shape, configuration or composition of pattern or color, or combination of pattern and color in three-dimensional form containing artistic value.

Summary of Module

- In this module you learnt the understanding of all the work and professional ethics that is necessary part of any project and the core demand of any organization. As your course is regarding the different learning projects and the team work so it will definitely help you understand the importance of performing the various tasks in groups and teams .To understand the project demand and requirements, making the different projects and realization of different skills or working on verity of projects will encourage you to take the different and strong decisions. No doubt taking decisions are tough tasks but the implantation on such decision and chose the right skill person to work on that is also an uphill task. Implementation of different decision either partially or impartially is also important factor. Through using different communication techniques and skills will help you to learn and work towards the success.

- You also learnt responsibilities of both employer and employee's, rules and regulation which should be follow in the workplace premises and property and confidential information protection.

Frequently Asked Questions (FAQS)

FAQ 1:	Do you know about the goodness?
Answer	The state or quality of being good in the society. Goodness is the nature and character of God.
FAQ 2:	What is trade secrets?
Answer	The Inner Bound of Proprietary Information
FAQ 3:	What is leave Encashment?
Answer	Encashment of leave means that employee will receive the cash for leaves which are not taken by the employees.
FAQ 4:	Develop a 2D drawing with Plot area of 30' x 45 '. What will be the "Gathering information" stage of life cycle methodology for the development of drawing?
Answer	Gathering information" stage of life cycle methodology as below The CAD Operator should queries regarding bedrooms, bathroom, porch etc. For example: Develop a 2D drawing to meet supplied floor plan with the following specifications: • Plot area 30' x 45' • 2 Bedrooms of dimension 12' x 15' with attached bath • Kitchen • lounge • stairs • Entry foyer
FAQ 5:	What is mean by guesstimate?
Answer	Make an estimate without complete information with guess work.
FAQ 6:	What is mean by estimate?
Answer	Estimate gives you a general idea about the value, size, or cost of (something) with strong information.
FAQ 7:	Importance of communication
Answer	Importance of communication has always been realized in all times because it is the most vital means by which people are connected together in the society to share information, ideas, without communication you will be unable to work in professional environment.

FAQ 8:	What is involved in the communication process?
Answer	Communication process involved 1. Idea 2. Encoding 3. Channel 4. Decoding 5. feedback
FAQ 9:	Level of Communication?
Answer	1. Intrapersonal Communication: When you think, you use intrapersonal communication 2. Interpersonal Communication: Conversation between two or more than two person to share ideas and information.
FAQ 10:	Do you know the symbol of copyright?
Answer	Yes! ©, For example.

Test Yourself!

Please mark the correct one from the given options. You can check your answer with the Answer Key at the end of this module.

1. Morality is?

a. Act of student in class
b. Goodness or badness of human Act
c. Follow Rules
d. None

2. The word *ethics* comes from_____.

a. Latin
b. Germen
c. Greek
d. None

3. To test yourself by_____.

a. Self-assessment
b. Other assessment
c. Self-examine
d. Both a and c

4. Which is/are the result of Business Abuse?

a Show Illegal act
b Show Product prices very high
c Unauthorized Discounts
d Both b and c

5. SOP Stand for?

a System Operating Procedure
b Standard Operating Process
c Standard Operating Procedure
d Standard Operation Procedure

Answer Key

MCQ Number	Correct Answer
1	b
2	c
3	d
4	c
5	c

AUTOCAD

Module-2

LEARNING GUIDE

National Vocational Certificate Level 3

Version 1 - July 2013

Module 2: Perform Orientation about AutoCAD 2D Fundamentals

Learning Outcomes

After completion of this learning module, you will be able to:

- Control the Displays in Drawings
- Create Basic Drawings
- Manipulate Objects as Desire

Learning Unit 1: Controlling Displays in Drawings

Overview

This unit describes fundamentals of AutoCAD 2D. After completion of this learning unit, you will be able to initialize the new file and save it. Moreover you will also be able to demonstrate visual reference commands like Zoom in Zoom out, PAN real-time and Status bar.

Remember

Ctrl + N For new File

Ctrl + S For Save the File

UN Command to Set the Unit of Drawing

1. Create or Open New File

Create or open means to get a new drawing in AutoCAD or learner can also use a template already available in CAD templates library.

Procedure

How to Open New File

Do you know?

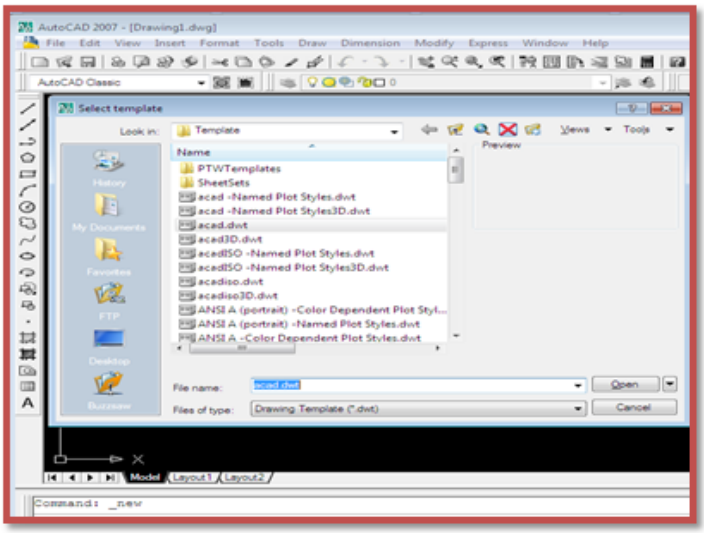
Templates: is a pre-define layout available in AutoCAD Software which assist the learner.

Learner can open AutoCAD file by two methods.

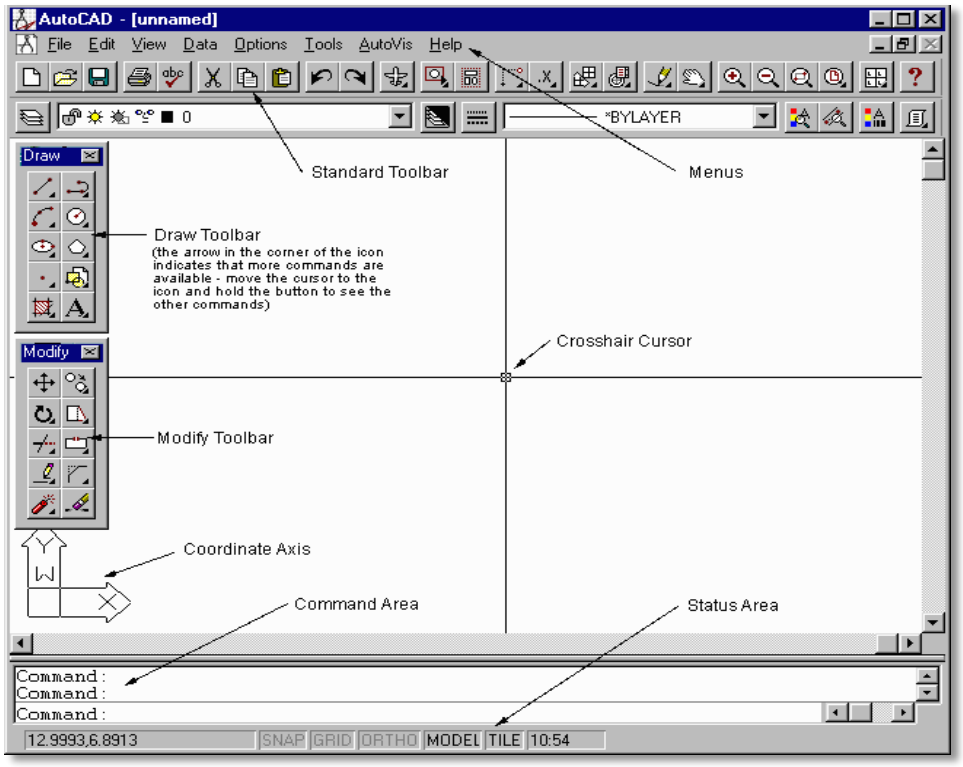
Method-1: Click Start Button→All Program→AutoDesk→Click AutoCAD (Lower/2013/higher Version)

Method-2: Double Click on the AutoCAD Icon at the Desktop

1.1 Create New Template: Learner can open or create a templates by clicking File→New→ **Select Template** as shown in the figure.



AutoCAD Interface



1.2 **Save the File:** Save the file before anything draw in it as a primary step to secure drawing.

There are three methods to save a file

Method-1: Through Short Cut Key: **Ctrl + S**

Method-2: From Standard Bar Click on

Method-3: Through File Menu: Click File→save then Name the file and Location where to save.

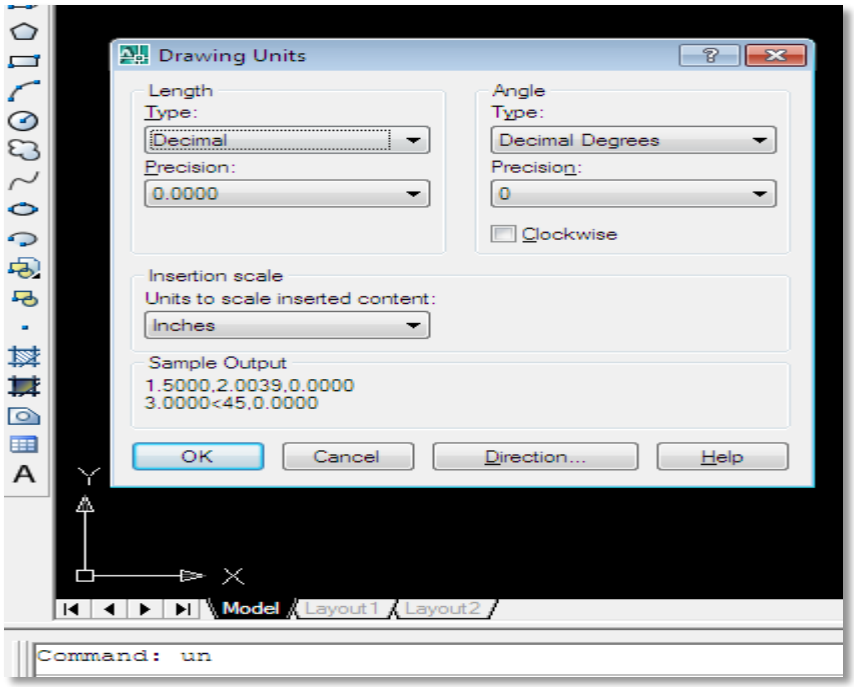
1.3 **Create Drawing**: Drawing is a basic communication in all engineering discipline like Architectural, Mechanical, Electrical etc. Each discipline has its own unit of drawing.

1.3.1 **Unit setting**: is the Primary step to set the unit according to the Discipline. Unit can be set by using **UN** command in command bar or from Main Manu through Format Tab.

Procedure: Format→Units…

Procedure

Write UN in command bar to get drawing units dialog box.



1.3.2 Unit Control

The dialogue box is separated into four main sections.

1) **Length type:** Which refers to linear units: Like Decimal, Architectural etc.

2) **Angles:** referring to angular units

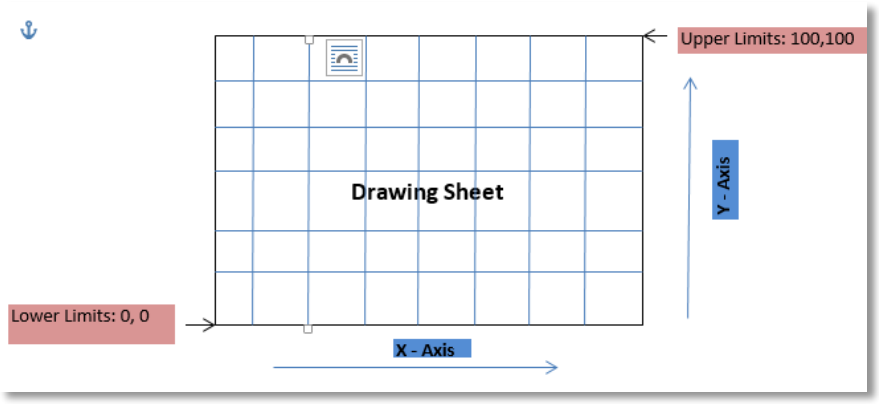
3) **Precision:** Precision is used to help a user to specify the precision value is to be needed for the drawing. Ideally precision value varies in terms of number of zeros after the decimal or fraction values. More Zero after decimal more precision will occur in drawing.

4) **Insertion Scale**: Selection of different measurement as per length type, like feet or inches for Architectural drawing.

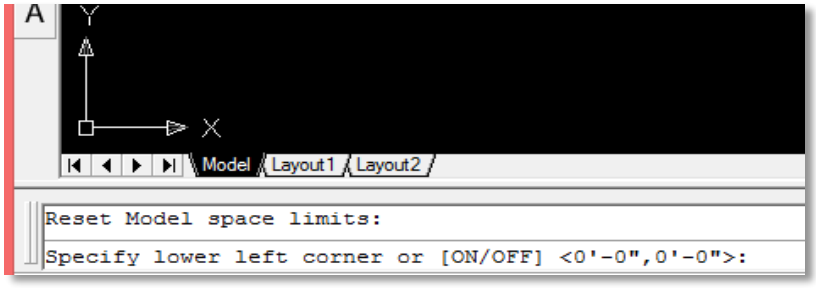
1.3.3 Status Bar: The Drawing Status Bar that displays the current display configuration of the drawing.

1.4 Drawing Limits: In AutoCAD, setting limits correctly lets you display the drawing grid over your working area, among other things. When you want to display all grids of limits you have to on Grid element from the status bar.

Drawing Limits define drawing sheet with horizontal and vertical coordinates as shown below.



Procedure

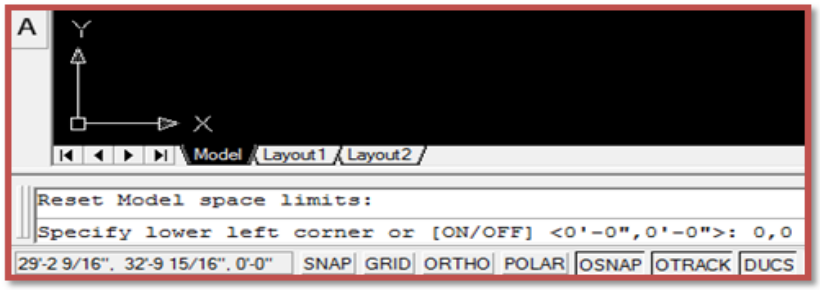


Step-1: Type **limits** command in Command bar, you will get the following Instructions.

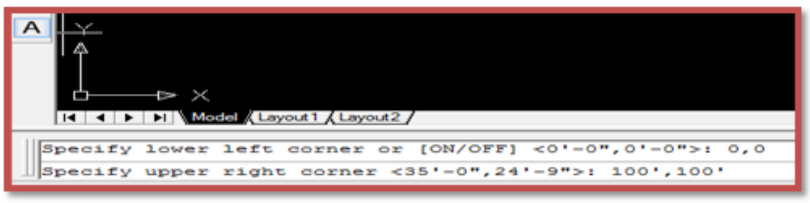
Step-2: Define the lower Left corner as 0, 0

Remember

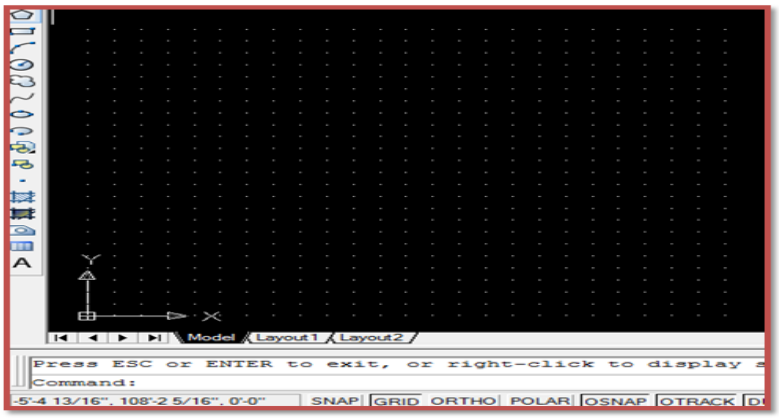
Limits can be adjusted according to Length type**.**



Step-3: Define the Upper Right corner as 100-feet, 100-feet.



Step-4: Grid Display: Now you can see the define limits to On Grid elements in status bar.



1.5 **PAN Real-time:** Pan is used to reposition the view dynamically with in the drawing area.

Procedure

Step-1: Select the **Pan-Real-time** option form the Standard Tool bar.

Step-2: Press Left Button from **Pointing device** and move the drawing dynamically in the drawing area.

Remember

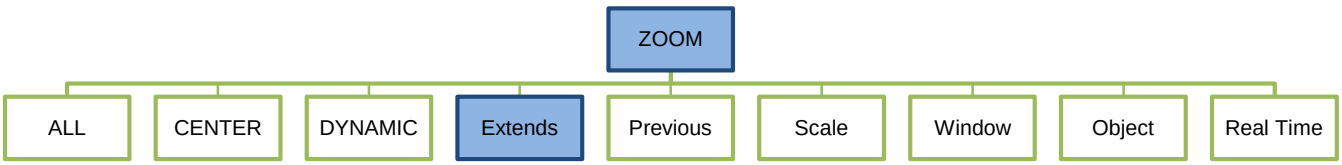
Command for Pan Real-Time:

Type P or PAN in Command bar.

Procedure

Press 3D/Scroll button of the Mouse and move the drawing.

1.6 **Zoom:** Zoom is used to Increases or decreases the apparent size of objects in the current View.



1.7.1 Zoom Extends: Zooms to display the drawing extents and results in the largest possible display of all the objects in the drawing area.

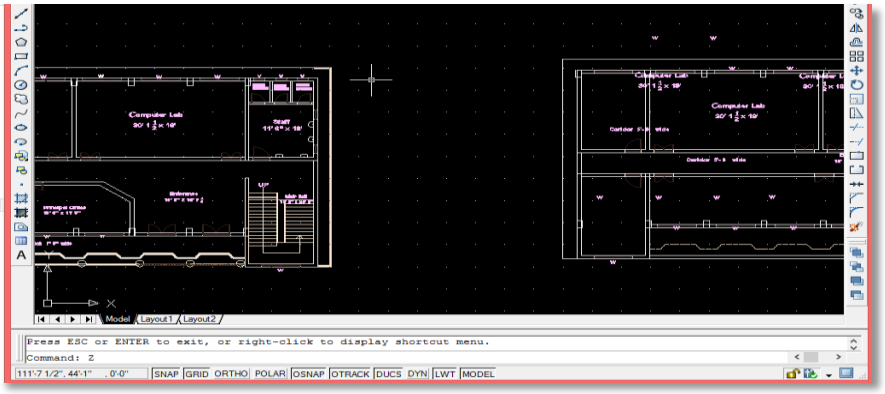
Procedure

Step-1: Type **Z** in command bar

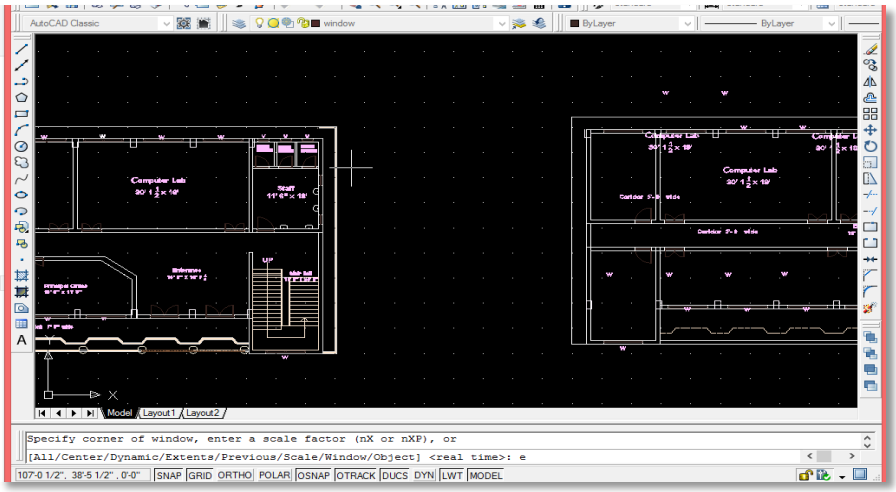
Remember

ZOOM Sub Command

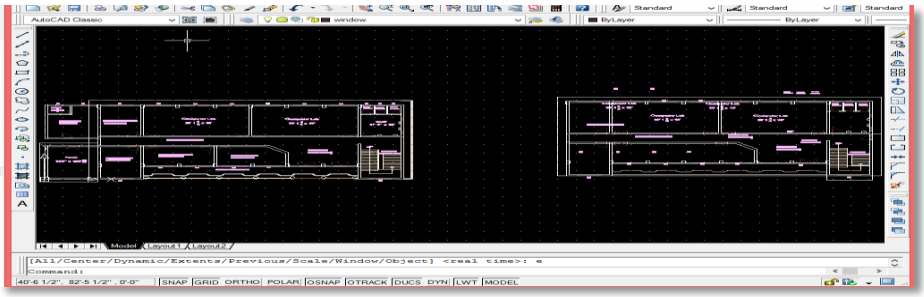
Sub Command of Zoom will be discuss in **Unit 3.1** in detail.



Step-2: Type **E** as sub Command



Step-3: Drawing extents and results in the largest possible display of all the objects.



Learning Unit 2: Create Basic Drawings

Overview

This unit teaches basic technique of making 2D drawing in AutoCAD, and let the learner to know about different coordinate systems (Absolut, Relative and Polar Coordinate system). After completion of this unit you will be able to draw objects by using line and circle commands and to know positions using the basic entry methods.

1. Coordinate System

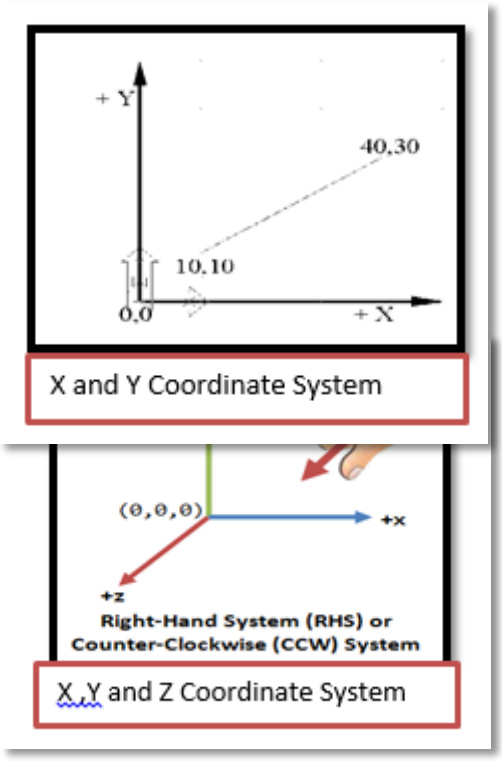
Real object can be view in 3 dimension (Width , Length and height), In AutoCAD these 3 dimension is represented with three Position X,Y and Z axis, is known as the Co-ordinate System.

There are two representation of coordinate dimension

1) 2-Dimension
2) 3-Dimension

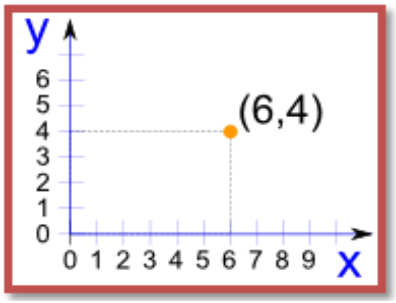
2-Dimension: A physical object in which length and width are real attributes. Any object can be draw with respect to x and y coordinates.

3-Dimension: A physical object in which length, width, and depth are real attributes—a model, with X , Y, and Z-axes that can be rotated for viewing from different angles**.**



2.1.1. World Coordinate System: All objects drawn in AutoCAD based on a simple X, Y or X, Y, Z co-ordinate system is known as the World Coordinate System (WC,).

2.1.2. Cartesian Coodinate System: A system which define the point draw in a Plan with respect to 2 Axis (X,Y) or 3 Axis (X,Y ,Z) in the same unit of length.



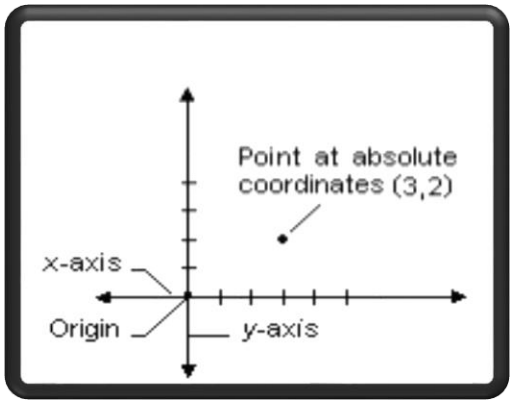
Point **(6,4)** is 6 units across (in the **x** direction), and 4 units up (in the **y** direction)So **(6,4)** means:

Go along 6 and then go up 4 then "plot the dot".

Example:

2.1.3. Entering Point Styles in AutoCAD: You can enter points directly on the command line using three different systems.

- Absolute Co-ordinate
- Relative Co-ordinate
- Polar Co-ordinate

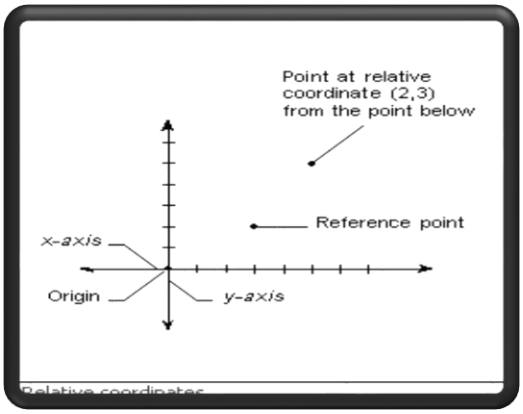


Absolute Coordinates: Coordinates that are defined in terms of their distance from the origin, the point where the axes intersect.

Syntax: X, Y

I.e. only give two values separated by comma.

Relative Coordinates: Coordinates that are defined in terms of their distance from a given starting point, rather than from the origin (intersection of two axes).



Syntax: @X, Y

Polar Coordinates: The fixed horizontal line in a system of polar coordinates from which the object is drawn by giving distance with angle.

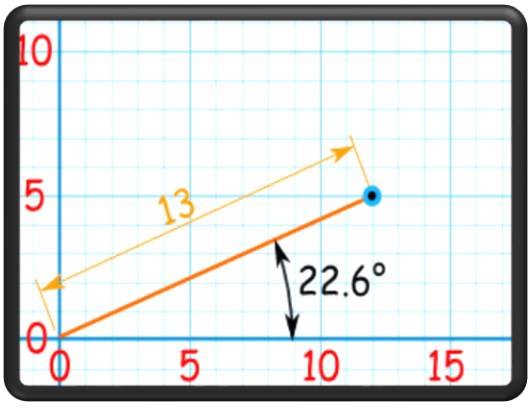
Syntax: @distance<angle

Where

Distance=length

< = (symbol)

Angle= to which direction the object is required.



In the Figure the line draw under the following command.

@13<22.6 Where Distance= 13, Angle=22.6

2.1.4. User Coordinate System (UCS)

User Coordinate System is a user define coordinates system that adjust a Plan for drawing and modeling. User can change the origin according to the object orientation.

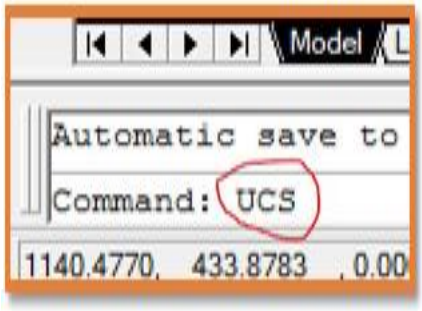
Procedure

UCS can define by two possible methods.

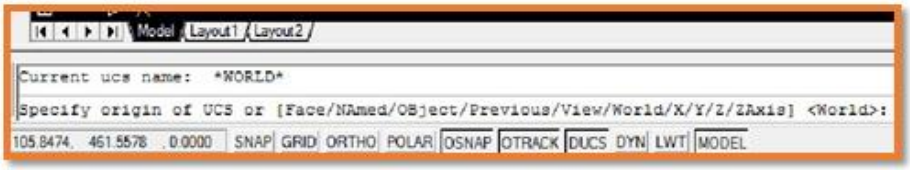
Remember

UCS will explain in more detail in **Module-4.**

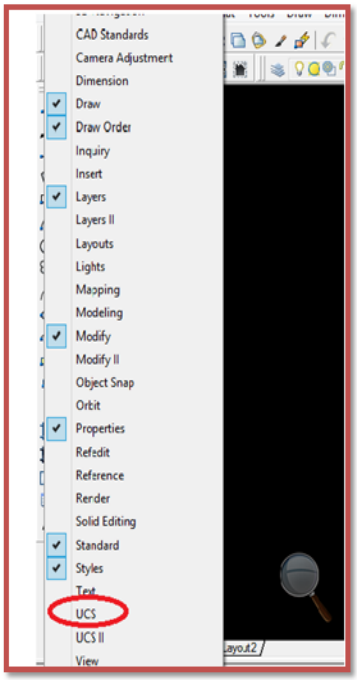
Method-1: Type **UCS** in command Bar to View all option of UCS as shown in Figure below:
Step-1: Type UCS



Step-2: You will see different UCS option.



Method-2: Right Click on the **Standard tool** bar and select the **UCS option** within the **POPUP Window.**



Step-3: You Will See **UCS Icon display**

Remember

Length Type = Architectural **and**

Unit to Scale = Feet/Inches

2.2. World Space :

There are two Space of drawing in AutoCAD. The normal space and the default are called **"World Space."** This is the space you will use almost all of the time. It is the normal 2-D or 3-D drawing with real world units.

The other mode of drawing or space is called **"Paper Space."** This space permits only a 2-dimensional world, and can be visualized as a flat sheet of paper.

Remember

Line (0, 0)

X Y

X, Y is a 2D Coordinates

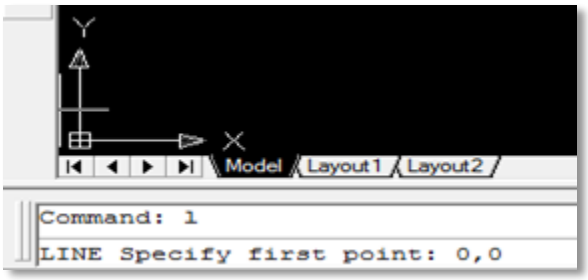
X,Y Define First point as an Origin

2.3. Unit Setup: Unit Setup is Primary step to prepare a Drawing sheet for a specific discipline as discussed above in **Unit 1.3.2**.

2.4. Line Command: Line Command is used to draw a line in Linear or Align direction.

Draw Manu	Draw Toolbar	Command Entry
Line		Line/L

Procedure



How to draw a line

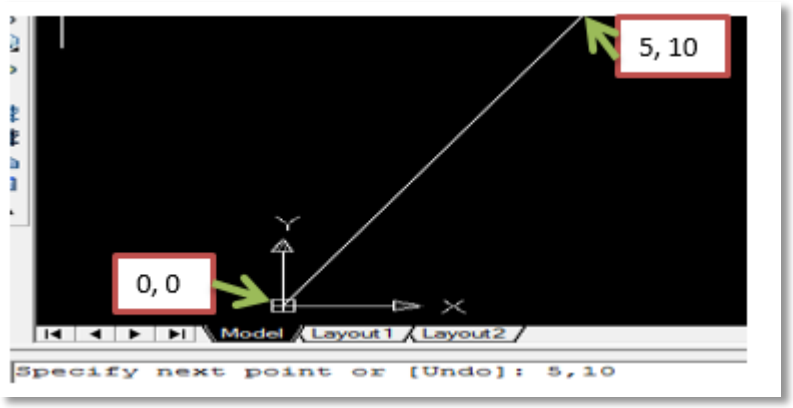
Step-1: Type L in command Bar
Step-2: Line Specify First Point: 0, 0 and Press Enter
Step-3: Line Specify Next Point: 5, 10 and Press Enter

Remember

Unit Indication

Foot = n' (Single Quote)

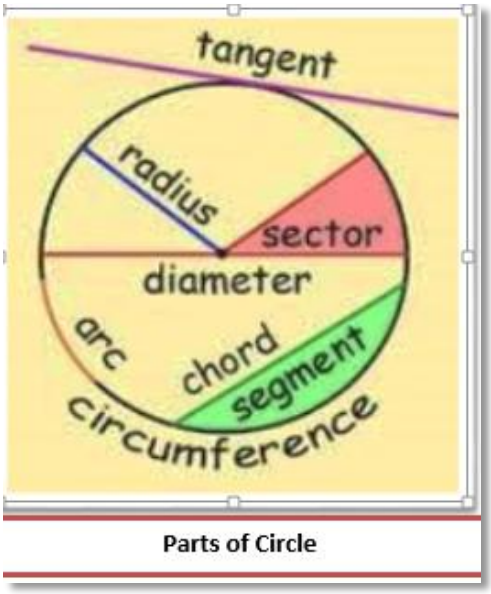
Inches =n"(double Quote)



Learner's Activity

- Draw a rectangle using line command having width = 10' and Length = 5'
- Draw an Equilateral, Isosceles and Right angle Triangle.

2.5. Circle: The figure formed by the group of points that are an equal distance from a point, or center. The angles of a *circle* add up to 360 degrees.



Parts of Circle

Parts of Circle

- **Circumference:** Length of circle is called circumference.
- **Radius:** A straight line from the center to the circumference of a circle
- **Diameter:** A straight line passing from side to side through the center of a body or figure, esp. a circle or sphere. .
- **Chord:** Any Straight Line which connects two points of circle is called chord, where as it does not passes from center of circle.
- **Tangent:** Any Straight Line which touches one point of circle is called Tangent.
- **Arc**: Part of a circle between any two points is called arc.
- **Sector:** Any area which is constructed by two Radiuses and an Arc is called Sector.
- **Segment:** Any area which is constructed by chord and an arc is called segment.

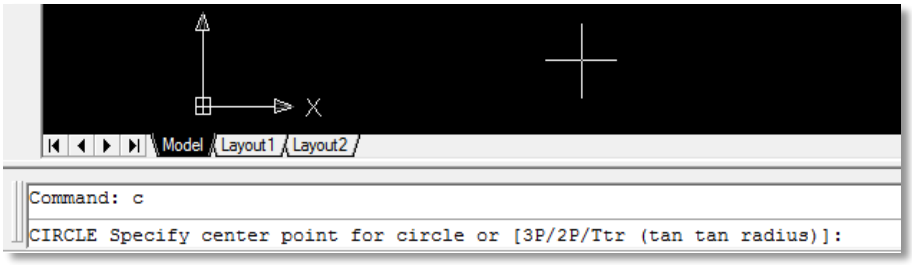
2.5.1. Circle Command

Procedure

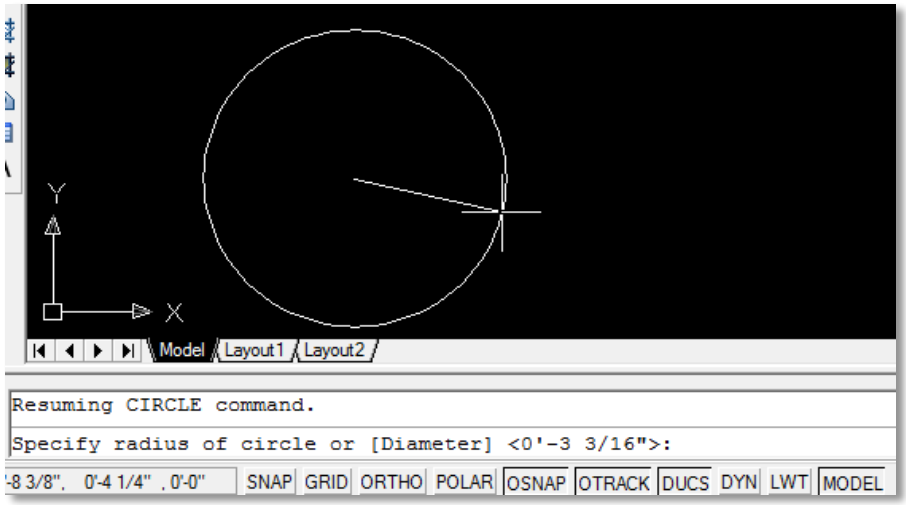
How to draw a Circle

Draw Manu	Draw Toolbar	Command Entry
Circle		Circle/C

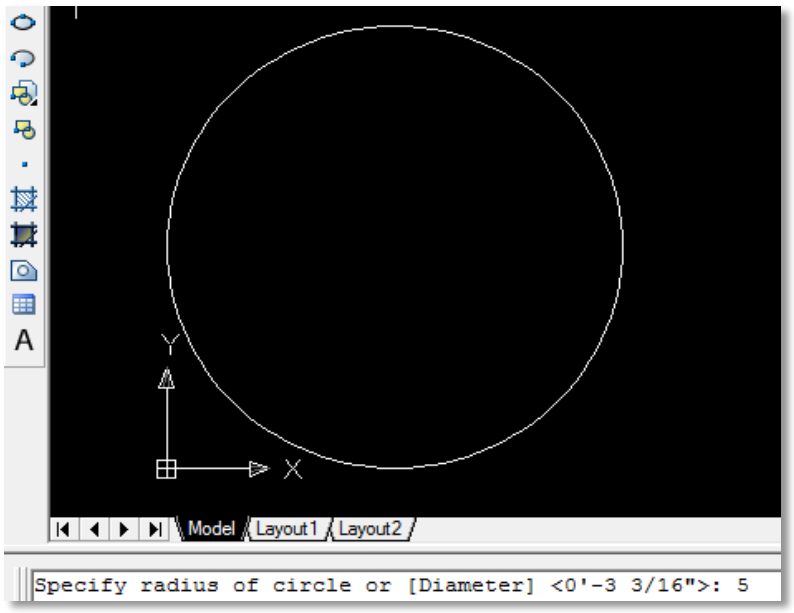
Step-1: Type C/Circle in command Bar and Press Enter



Step-2: Specify any Point for Centre I.e. **5, 5** and Press Enter



Step-3: Specify radius of Circle or Diameter: **5** and Press Enter.



2.5.2 Tan Tan Radius (TTR):

Procedure

How to draw a Circle with TTR

Step-1: Type **C** and press ENTER (You See: 3P/2P/TTr)

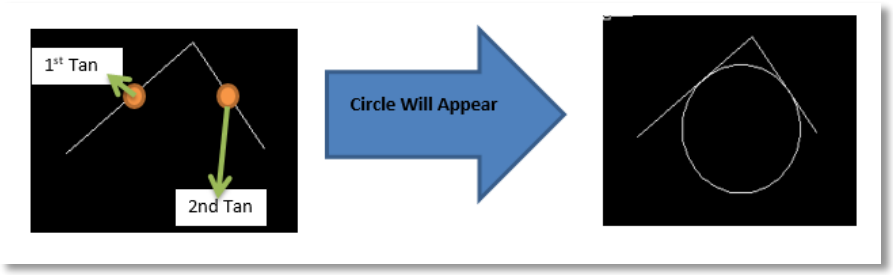
Step-2: Type **T** for (Tan Tan Radius (TTR))

Step-3: Specify point on object for first tangent of circle: Click on the

First Tangent Element (Where the circle will touch)

Step-4: Specify point on object for second tangent of circle: Click on the Second Tangent Element (Where the circle will touch)

Step-5: Specify radius of circle <0'-0>: 2



2.5.3 *3-Points*:

Procedure

How to draw a Circle with 3-P

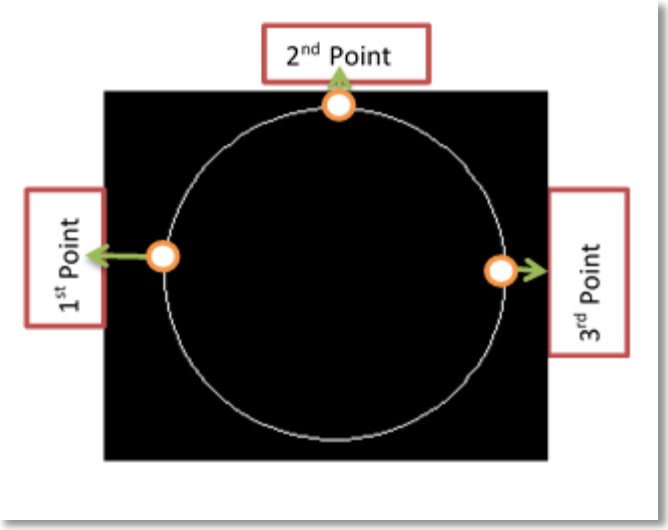
Step-1: Type C Command (see methods above)

Step-2: 3P/2P/TTR/<Center point>: Type **3p** and then press.

Step-3: Specify first point on circle: Click on First Point.

Step-4: Specify second point on circle: Click on second Point.

Step-5: Specify third point on circle: Click on third Point.



2.5.4 *2-Points*

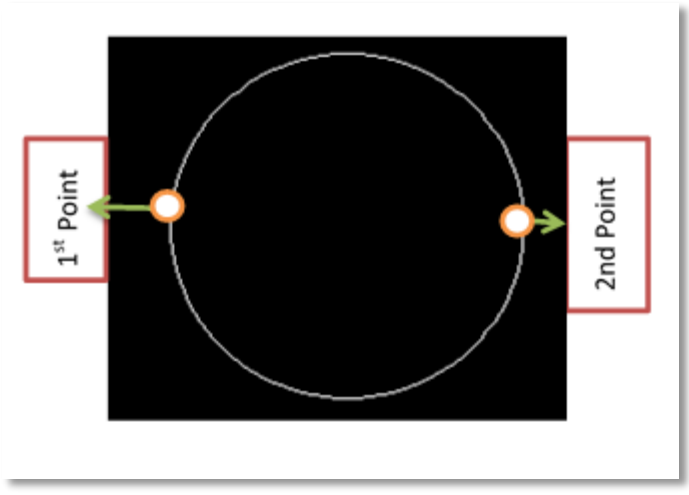
Procedure

How to draw a Circle with 2-P

Step-1: Type C Command (see methods above)
Step-2: 3P/2P/TTR/<Center point>: Type **2P** and then press.

Step-3: Specify first end point of circle's diameter: Click on First Point

Step-4: Specify second end point of circle's diameter: Click on second point.



2.5.5 Tan Tan Tan (TTT): Another sub command of Circle you can activate from the Draw Manu: **Draw→circle→TTT**

Procedure

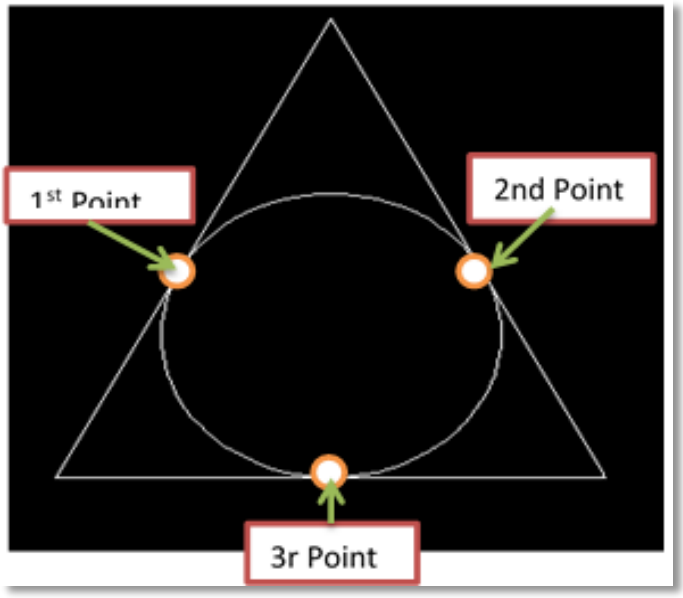
How to draw a Circle with TTT

Step-1: Click on Draw Manu: **Draw→Circle→TTT**

Step-2: 3p Specify first point on circle: _tan to. First Point

Step-3: 3p Specify second point on circle: _tan to. Second Point

Step-4: 3p Specify third point on circle: _tan to. Third Point



Learner's Activity

- Draw a circle with radius of 5"
- Calculate a diameter if Radius is 10"
- Draw a circle with 3-P touching outer corner of Equilateral Triangle.

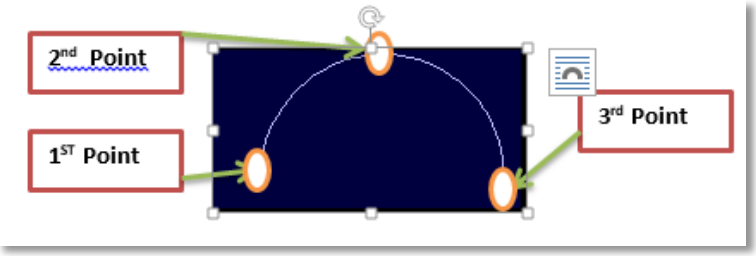
2.5.2. Arc Command: Arc is used to draw an arc between two lines from one point to the next**.**

Draw Manu	Draw Toolbar	Command Entry
Arc		Arc/A

Procedure

How to draw an ARC

Step-1: Type **ARC/A** in command bar and Press **Enter**
Step-2: ARC Specify start point of arc or [Center]: **Click Start Point**
Step-3: Specify second point of arc or [Center/End]: **Click second Point**
Step-4: Specify end point of arc: **Click end Point**



2.6. Format: Format option is used to change properties of an object by defining Colour of an object, by layer option , by Line type ,line weight by changing the Units as per discipline as discussed previously.

2.7. SNAP Option: Snap mode restricts the movement of the crosshairs to intervals that you define. When snap mode is on, the cursor seems to adhere, or "snap," to an invisible rectangular grid. Snap is useful for specifying precise points with the arrow keys or the pointing device.Grid mode and Snap mode are independent but are often turned on at the same time.

Learning Unit 3: Objects Manipulation

Overview

This unit teaches Modification in 2D drawing, and let the leaner to know the manipulation of an object as desire. After completion of this unit you will be able to manipulate erase, delete, and Zoom objects.

1. Use the Erase Command

3.1 ERASE Command: Erase command is used to removes objects from a drawing.

Modify Manu	Modify Toolbar	Command Entry
Erase		Erase/E

Procedure

How to Erase an Object

Step-1: Type **Erase/E** in command bar and Press Enter

Step-2: Select Object and Press **Enter**

The Selected Object should be Erase/Delete.

Do You Know!

Alternate Method

Select the object/s and Press **Delete** button

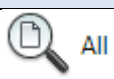
Learner's Activity

- Use Erase command to delete a circle previously created in Learner Activity -2

3.2 Zoom: Increases or decreases the apparent size of objects in the current view. Zoom is divided in different sub command already discussed in **Unit 1.7.**

Detail of Zoom Sub Command:

3.2.1 ZOOM→ All: This is used to display the whole drawing as far as its drawing limits or drawing extents.

View Manu	Icon	Command Entry
Zoom→All		Z→A

3.2.2 ZOOM→Center: This option is used to display a window defined by a center point and a magnification value or height. A smaller value for the height increases the magnification. A larger value decreases the magnification.

View Manu	Icon	Command Entry
Zoom→Center	Center	Z→C

3.2.3 ZOOM→Dynamic: Zooms to display the generated portion of the drawing with a view box. The view box represents your viewport

View Manu	Icon	Command Entry
Zoom→Dynamic	Dynamic	Z→C

3.2.4 ZOOM→Previous: This option is used to display previous view; it can restore up to 10 previous views.

View Manu	Icon	Command Entry
Zoom→Previous	Previous	Z→P

3.2.5 ZOOM→Scale: It simply requires the entry of a number that represents a magnification factor.

View Manu	Icon	Command Entry
Zoom→Scale	Scale	Z→S

3.2.6 ZOOM→ Window: One of the types of Zooms to display an area specified by two opposite corners of a rectangular window.

View Manu	Icon	Command Entry
Zoom→Windows	Window	Z→W

Procedure

Step-1: Type Z Command in Command bar and Press Enter

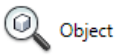
Step-2: Type **W** for Window option

Step-3: Select first corner

Step-4: select opposite corner

Step-5: Press **ENTER** to Zoom the Object within the Selected Window Area.

3.2.7 ZOOM→ Object: This option is used to display the selected object.

View Manu	Icon	Command Entry
Zoom→Object		Z→O

Summary of Module

- In this module we learnt about the fundamentals of 2D, how to open, create and save a file in AutoCAD. We also learnt the creating of primary commands like line, Arc, rectangle and circle in different ways with different coordinate System (Absolut, Relative and Polar etc.) to enhance the knowledge of geometric entities with the help of World Space.

- The module also described manipulation of different objects and drawing viewports with different Zooming options along with its procedures.

Frequently Asked Questions (FAQs)

FAQ 1:	Do you know about the Provider of AutoCAD?
Answer	Autodesk is the Provider of AutoCAD
FAQ 2:	Is AutoCAD limited to 2 Dimension Drawing?
Answer	No! AutoCAD Provides both environment for 2-D and 3-D as well.
FAQ 3:	What is the extension of CAD File?
Answer	.DWG is file extension
FAQ 4:	Is there any Pick Point in Line?
Answer	Yes! There are Three Points to Pick a line i.e. Start , Mid and End Point
FAQ 5:	Is Manual Drafting more reliable than AutoCAD?
Answer	No! AutoCAD is more reliable than Manual Drafting as because you can easily make modification in CAD drawing.
FAQ 6:	Can lower version of AutoCAD be opened in Higher version and Vise verse?
Answer	Yes! Lower version of AutoCAD can easily be open in higher version but higher version cannot be open in lower version.
FAQ 7:	Is there any way to drawn a circle other than by defining radius?
Answer	Yes! Circle can be drawn by giving the diameter of circle.
FAQ 8:	Can an object be moved within the drawing area?
Answer	Yes! It can move an object by apply MOVE command.
FAQ 9:	Can a file other than CAD drawing be opened in AutoCAD?
Answer	Yes!
FAQ 10:	Erase is the only way to delete an object/s?
Answer	No! You can use delete button to delete selected object/s.

Test Yourself!

Please mark the correct one from the given options. You can check your answer with the Answer Key at the end of this module.

1. What should you pay attention to when learning AutoCAD?

a. The Command Line
b. Status Bar
c. The Title Bar
d. Tool Icons

2. Polar coordinates are used mostly for drawing...?

a. Circles
b. Arcs
c. Vertical lines
d. Angled Lines

3. Which keystrokes will UNDO a command_____.

a. Control+U
b. Alt+U
c. Control+Z
d. Alt+Z

4. When drawing in 2D, what axis do you NOT work with?

a. X-Axis
b. Y-Axis
c. Z-Axis
d. WCS

5. What does WCS stand for?

a. Word Coordinate System
b. World Coordinate System
c. Watch Coordinate System
d. Non

Answer Key

MCQ Number	Correct Answer
1	a
2	d
3	c
4	c
5	b

AUTOCAD

Module-3

LEARNING GUIDE

National Vocational Certificate Level 3

Version 1 - July 2013

Module 3: Create 3D Interface Drawings

Learning Outcomes

After completion of this learning module, you will be able to:

- Develop Familiarity with 3D Basics Interface
- Introduce Thickness and Elevation
- Visualize the Model

Learning Unit 1: Develop Familiarity with 3D Basics Interface

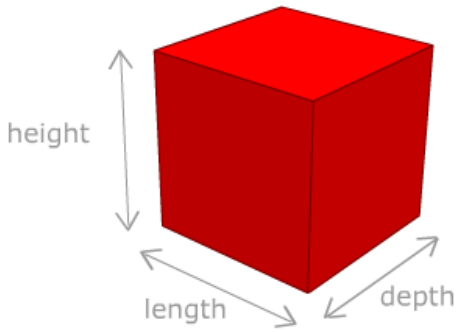
Overview

This unit describes familiarity with AutoCAD 3D basic interface. After completion of this learning unit, you will be able to work with different workspaces like 3D modeling and 3D Basic along with its Ribbons and Pull down menu. Viewports, Named Views, Steering Wheel and plan view techniques are the main functions are also discussed in this unit.

Further you will be able to apply extrude, sweep and loft commands to make object in 3D solid form with different views.

1. Introduction to 3D

Any object that has height, width and depth is known as 3D, like any object in the real world.



Remember

Some of the 3D shapes will be discuss in module 8.

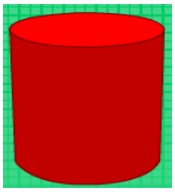
3D Shapes:

Some 3D objects:

1. Sphere

An Object shaped like a round ball. Every point on the surface of a sphere is at an equal distance from its center.

2. Cylinder



A solid tube with long straight sides and two equal sized circular ends

3. Cone

A solid shape with a round base which narrow to a point.

Your ability in using the basic drawing, modifying, and display commands is critical to your successful study of 3D for two reasons:

1. Drawing in 3D can seem more complex and difficult than drawing in 2D, and it can be frustrating until you get used to it. If you aren't familiar with the basic commands, you'll become that much more frustrated.
2. Accuracy is critical in 3D drawing. The effect of errors is compounded, so you must be in the habit of using tools, such as the **OSNAP** modes, to maximize your precision.

2. 3D Basics Interface

The following is AutoCAD's 3D Basic interface. The 3D Basic ribbons are as follows:

Remember

All of the 3D Basic will not covered in this module, some of 3D basic will be covered in later modules

- Create
- Edit
- Draw
- Modify
- Selection
- Coordinates
- Layers and Views

The 3D Basic pull-down menus are as follows:

- Home
- Render
- Insert
- Manage
- Output
- Plug-ins
- Online
- Express Tools

3. Setting UP the 3D Workspace

So far, you've been using the Drafting & Annotation workspace to create your drawings in 2D. Your first task in transitioning into 3D begins by switching to a new workspace for working in 3D and changing how AutoCAD displays the available tools.

Workspaces:

AutoCAD workspaces are sets of menus, toolbars and dockable windows (such as the Properties palette, Design Center, and the Tool palettes window) that are grouped and organized so that you can work in a custom, task-oriented drawing environment.

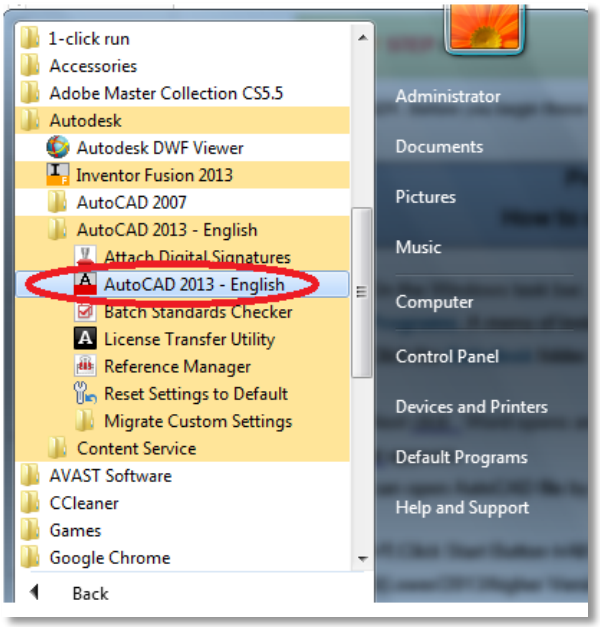
STEP BY STEP Operation

GET READY. Before you begin these steps, be sure to turn on and/or log on to your computer.

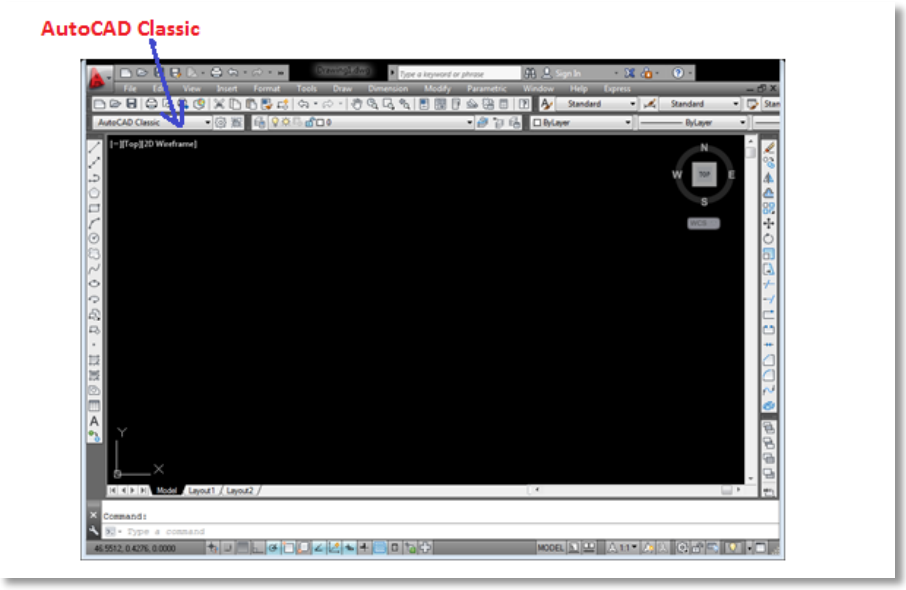
Procedure

How to set 3D Workspace

1. On the Windows task bar, click the **Start** button, then click **All Programs**. A menu of installed programs appears.
2. Click the **Autodesk** folder→ Next AutoCAD2013-English

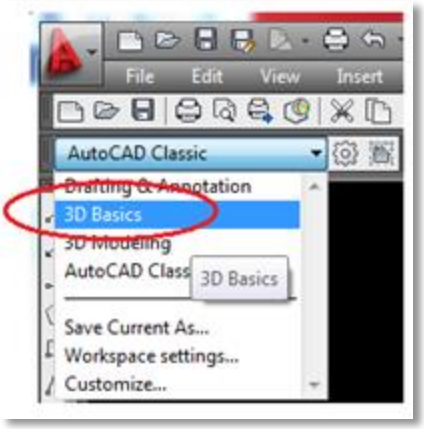


3. AutoCAD opens and a new blank drawing appears.

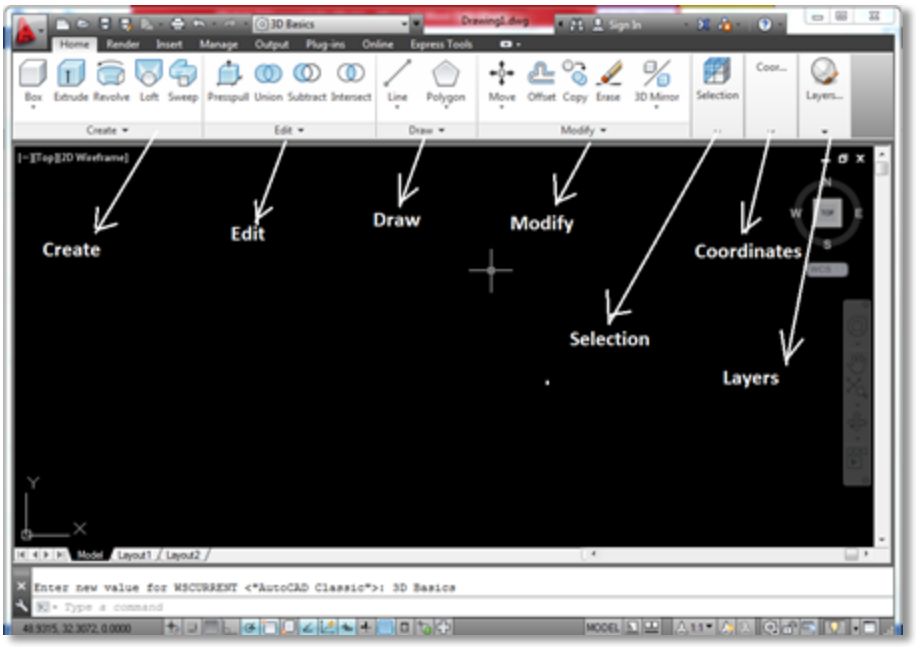


4. Click the workspace switching drop down arrow or icon.

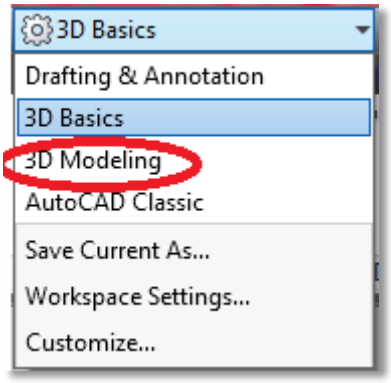
5. Click 3D Basic



6. The following figure show3D Basic Ribbons.



7. If you Click the 3D Modeling



4. 3D Modeling Identification

The 3D Modeling panels are as follows:

- Modeling
- Mesh
- Solid Editing
- Draw
- Modify
- Section
- Coordinates
- View
- Selection
- Layers and Groups

The 3D Modeling pull-down menus are as follows:

Note: Some of the 3D Basic, 3D Modeling Panels and Pull-down menus will manipulate in this module while some will be manipulated in later modules.

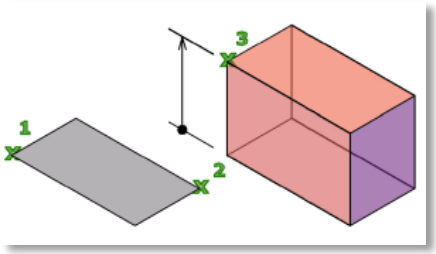
- Home
- Solid
- Surfaces
- Mesh
- Render
- Parametric
- Insert
- Annotate
- View
- Manage
- Output
- Plug-ins
- Online
- and Express Tools

STEP BY STEP AutoCAD

GET READY. Before you begin these steps, be sure set unit for architectural designing.

5. Working with 3D Basics

In this lesson you will learn 3D basic interfaces as discussed previously.



1. From Home Menu
Create→Box

Box: Box is a solid object based on Two Points and a Height
The base of the box is always drawn parallel to the *XY* plane of the current UCS (work plane). The height of the box is specified in the *Z*-axis direction. You can enter both positive and negative values for the height.

Home tab	**Button**	**Command Entry**
Box		box

Procedure
How to draw a Box

When you enter box command it will display the following list

First Corner

Starts the box by setting the first corner.

Other Corner

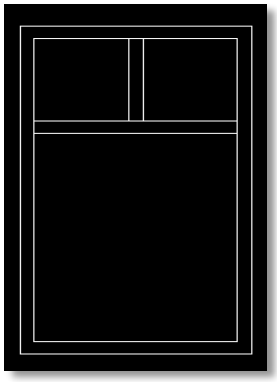
Sets the opposite corner and height of the base of the box.

Cube

Creates a box with sides of equal length.

Length

Creates a box with length, width, and height values you specify. The length corresponds to the *X* axis, the width to the *Y* axis, and the height to the *Z* axis.



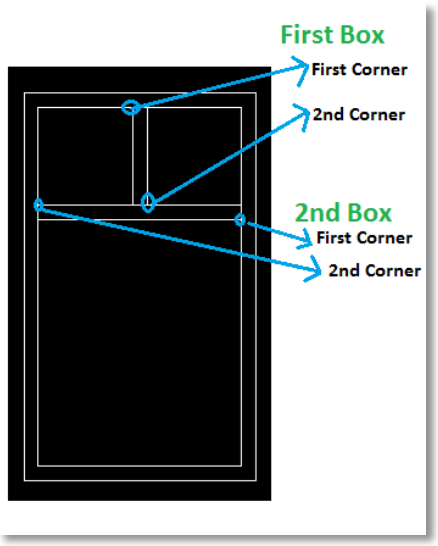
You are developing a 3D drawing for the following 2D drawing by using box command.

a. Enter Box Command or Click the Box Button
b. Specify the first corner of the base.
c. Specify the opposite corner of the base.
d. Specify the height up to 11 feet.

Now you'll use the in-canvas viewport controls to change the view from a plan view of the drawing—looking straight down at it—to one in which you're looking down at it from an angle. There are several preselected viewpoints, and here you'll switch to one of them. The in-canvas viewport controls are composed of three bracketed menus in the upper-left corner of the drawing window. Starting from the left, these menus are as follows:

- Viewport Control
- View Controls
- Visual Style Contro

Will discuss later in

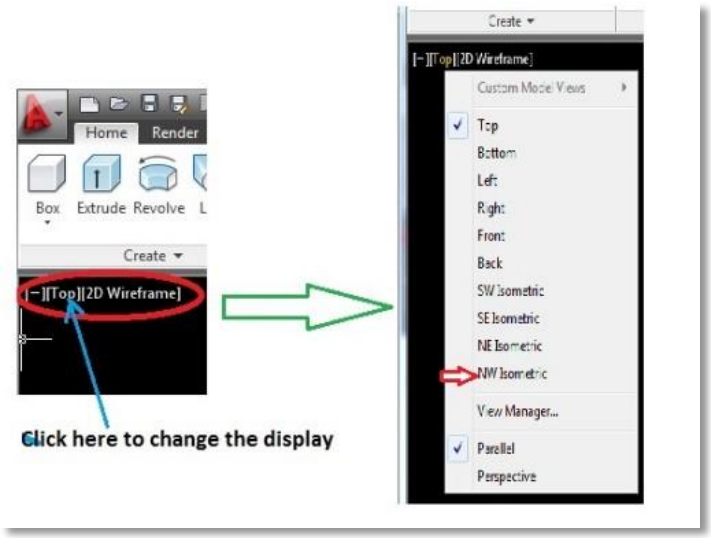


You'll use the View Controls menu to change the angle in which the drawing is displayed.

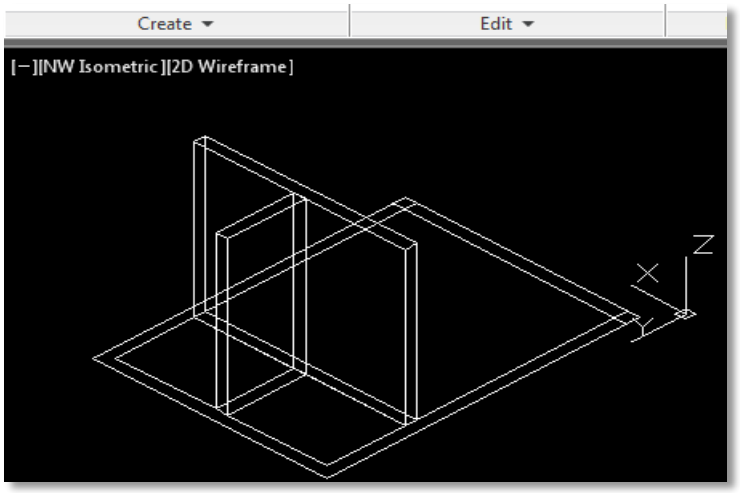
e. Using the in-canvas viewport controls, click the View Controls menu, currently displayed as [Top], and choose SW Isometric, as shown in figure below

Do you know!

You can use all options one by one to check the different views.



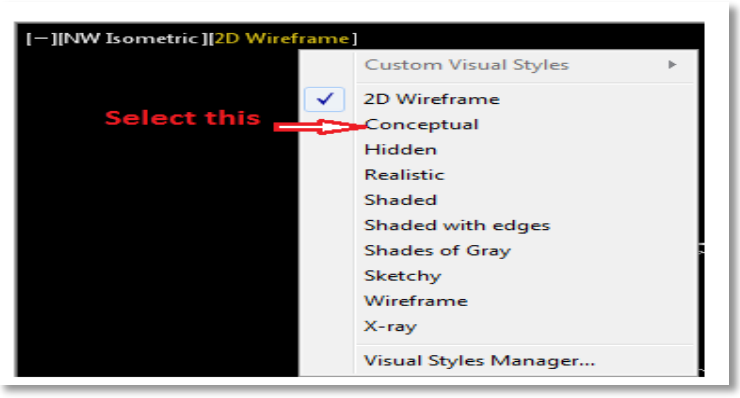
f. The view change to look like shown in the figure below



Do you know!

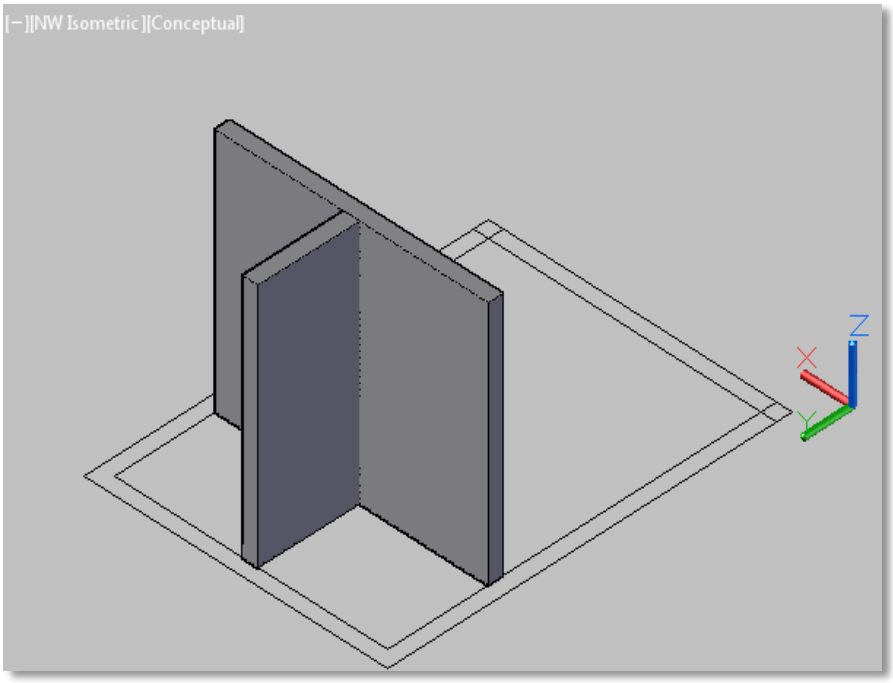
You can use all options one by one to check the different Visual Style

2D Coordinates are changed into 3D coordinates, x, y and z. X and Y arrows are still show the front view while Z shows the height of the object/s.



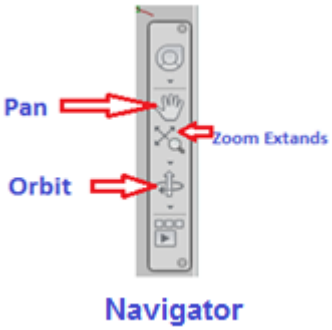
g. Again using the in-canvas viewport controls, click the Visual style Controls menu, currently displayed as [2D Wireframe], and choose Conceptual view, as shown in figure below

h. The visual style change look like shown in the figure below



TIP: Interesting:-

1. **Orbit:** Rotate the views in 3D.
2. **Pan:** Move the view planer to the screen.
3. **Zoom Extends:** Zoom to display the maximum extends of all



Learner's Activity

Q: Draw box by using cube and length options.

Create→Extrude

Extrude: extrude is used to create a solid object from circles, polygons, and objects, created from or converted to Polylines, that have "closed" paths**.**

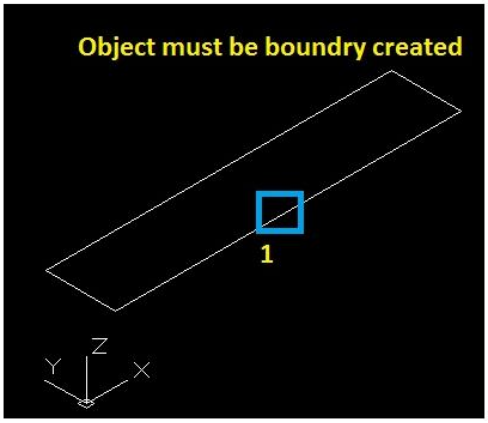
You can extrude an object by giving height to create a 3D surface or solid.

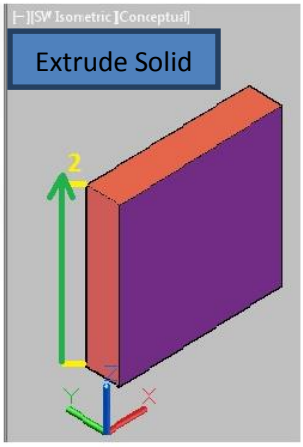
Home tab	**Button**	**Command Entry**
Extrude		Ext

Procedure

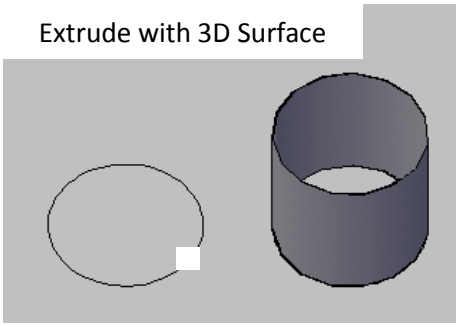
Apply Extrude command

1. **Extrude with solid**
   a. Draw rectangle
   b. Type **ext** in command line
   c. Select the object
   d. Define the height and press enter

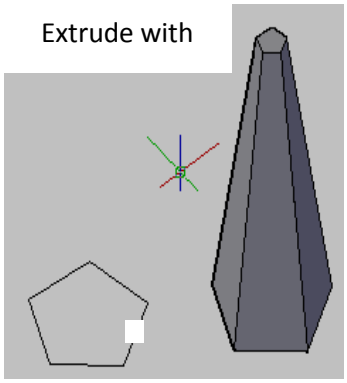




2. **Extrude with 3D surface**
   a. Draw a circle
   b. Type **ext** in command line
   c. Type **MO** for mode change
   d. Type **SU** for **surfac**e mode
   e. Select the object
   f. Press enter
   g. Define the height and press enter



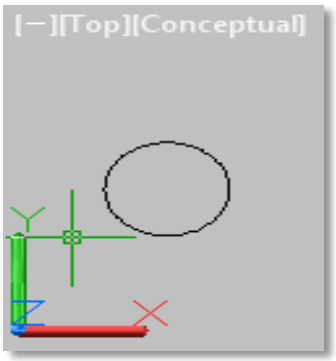
3. **Extrude with taper**
   a. Draw a polygon or something else
   b. Type **ext** in command line
   c. Select the object
   d. Type **T** for taper
   e. Define angle i.e. 10
   f. Define height i.e. 5
   g. Press enter



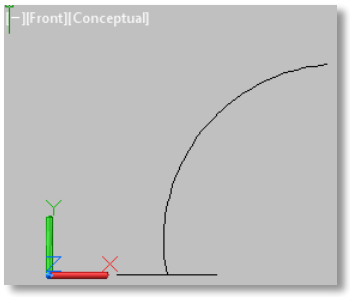
Specifies the taper between -90 and + 90 degrees

4. **Extrude with Path**
   Specifies the extrusion path based on a selected object. The path is moved to the centroid of the profile. Then the profile of the selected object is extruded along the chosen path to create solids or surfaces.

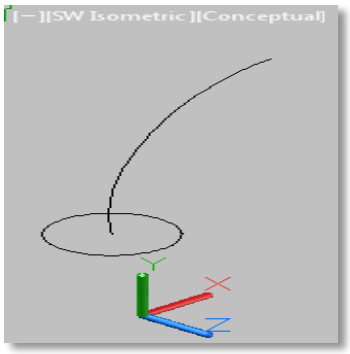
   a. Draw a circle in TOP view (Define any Radius/Diameter)



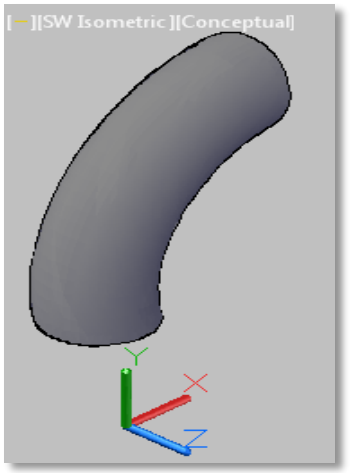
   b. Change the view from top to front and draw an arc



c. Change the view to SW Isometric



d. Apply Extrude: type **ext** in command line
e. Select the circle and press enter
f. Type **P** for path
g. Select the arc line and press enter, You will get the following



h. Try it in all view and enjoy the output.

Create→Sweep

Similar to the Extrude tool, the Sweep tool extrudes a 2D shape along a Path to create a 3D shape.

Create→Revolve

Revolve: Creates 3D solid or surface by sweeping an object around an axis.

There are two mode of operation perform by revolve

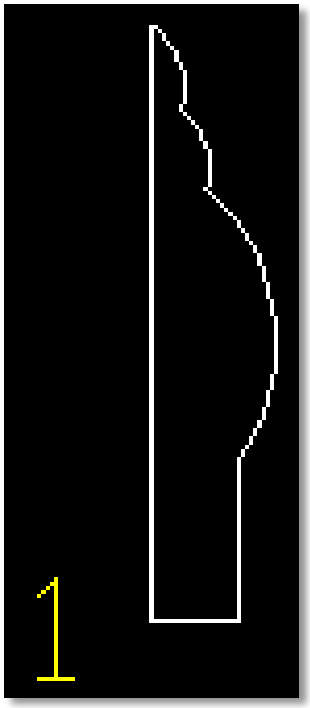
- Revolve: It creates a solid revolving object.
- Revolve surface: It creates a surface revolving object.

Revolve

Home tab	**Button**	**Command Entry**
Revolve		rev

Procedure
Apply Revolve command

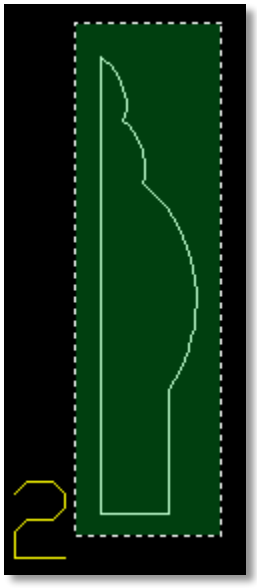
Apply revolve command to following object.



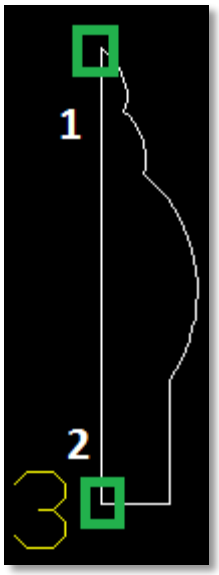
Step-1: Draw the object as shown in the figure-1.

Step-2: Type **rev** in command bar or click the **revolve** button.

Step-3: Select the object either as a whole or as single if 2D polyline converted.

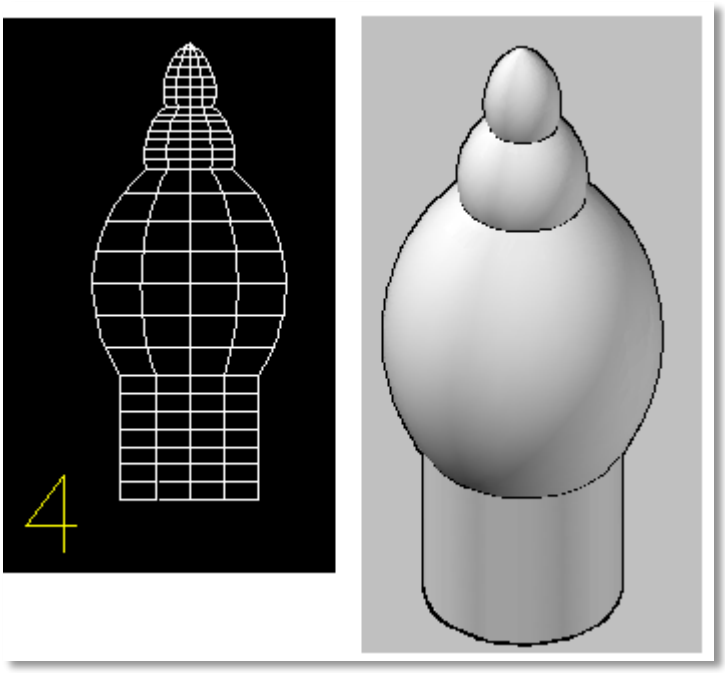


Step-4: Select object that defines the axis of revolution: <pick the vertical line>



Step-4: Specify angle of revolution <360> : <ENTER>

In result you should get the Following



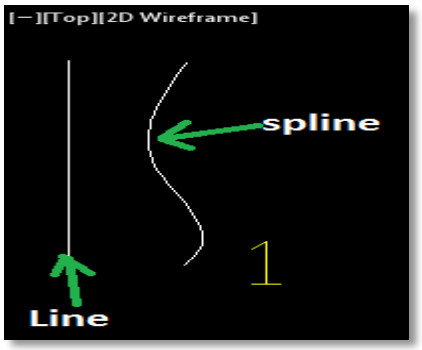
Revolve Surface

Home tab	**Button**	**Command Entry**
Revolve		revsurf

Procedure

Apply Revolve surface command

Apply revolve surface command to following object.



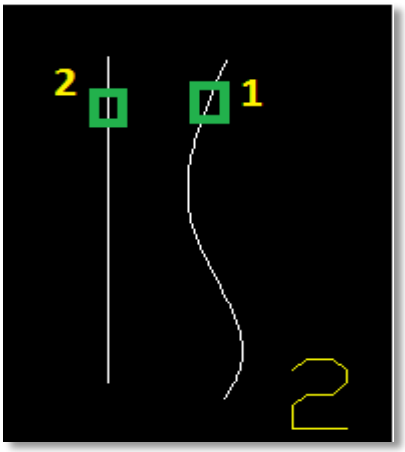
Step-1: Draw the object as shown in the figure-1.

Step-2: Type **revsurf** in command bar or click the **revolve** button→ type **MO→SU**.

Step-3: Select the object to revolve and press Enter as shown in figure below.

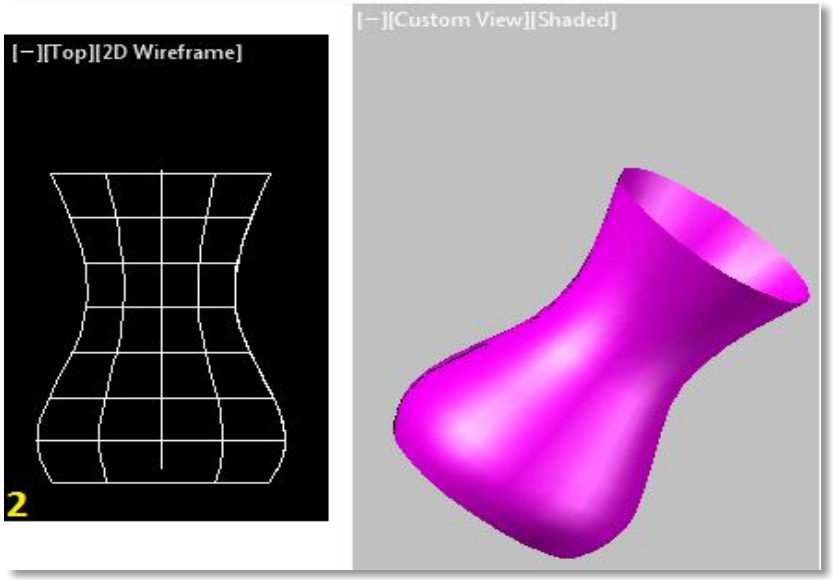
Step-4: Type **O** in command bar

Step-5: Select an object



Step-6: Specify angle of revolution <360> : <ENTER>

In result you should get the Following



Create→Loft

Loft: Creates a 3D solid or surface in the space between several cross sections.

LOFT draws a solid or surface in the space between the cross sections. You must specify at least two cross sections when you use the LOFT command.

You can use the following objects and sub-objects with LOFT:

Objects That Can Be Used as Cross Sections	Objects That Can Be Used as a Loft Path	Objects That Can Be Used as Guides
2D polyline	Spline	2D spline
2D spline	Helix	3D spline
Arc	Arc	Arc
Circle	Circle	2D polyline **Note:** 2D polylines can be used as guides if they contain only 1 segment.

Home tab	Button	Command Entry
Loft		loft

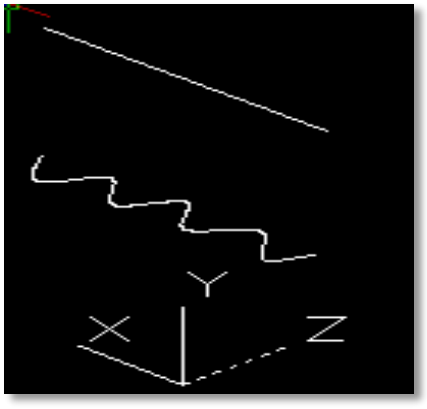
Procedure

Apply Loft command

Step-1: Draw spline in top view.

Step-2: Draw a straight line on the top of spline in front view.

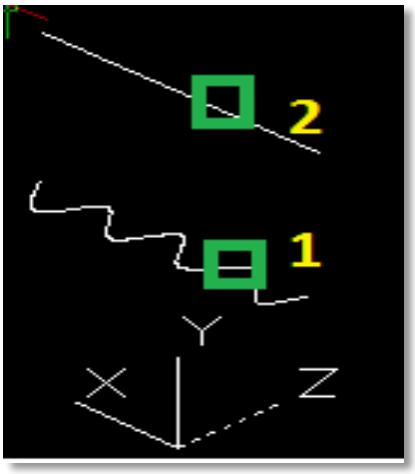
Step-3: Change the view into Isometric, you should see look like



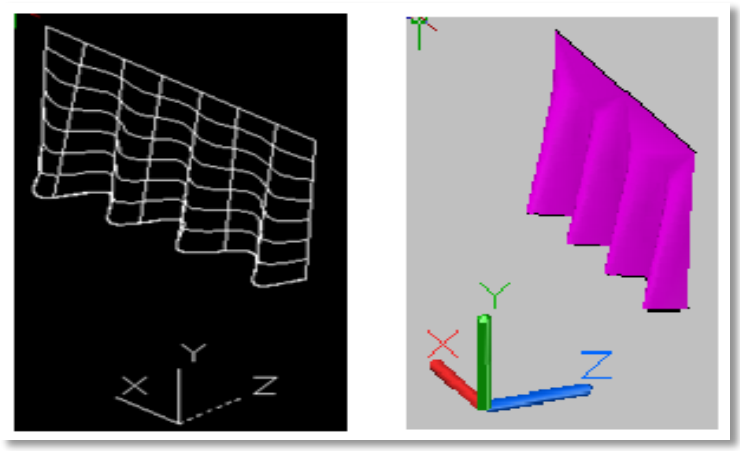
Step-4: Type loft command in command bar or click on loft button

Step-5: Select cross sections in lofting order

Step-6: Select cross sections in lofting order and press two time Enter



Step-7: You should get the following



Project-1

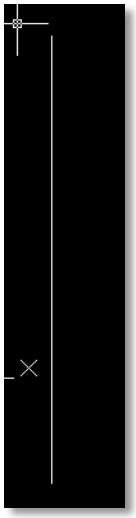
Making a Lamp

This project shows a complete procedure to draw a lamp, composites of **line, arc, offset, extrude, polygon, revolve and loft** 3D commands.

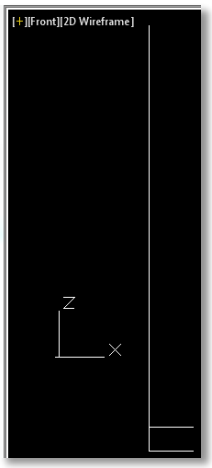
STEP BY STEP Operations on AutoCAD

GET READY. Before you begin these steps, be sure set unit for architectural designing.

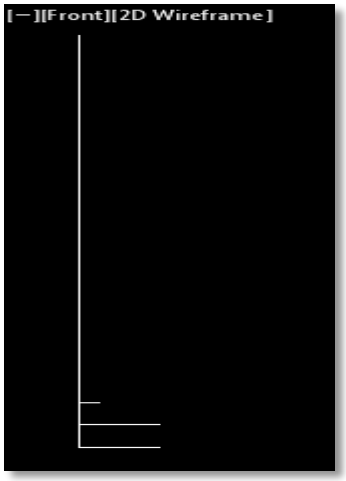
Step-1: Draw a line of 1.5 feet in Front View



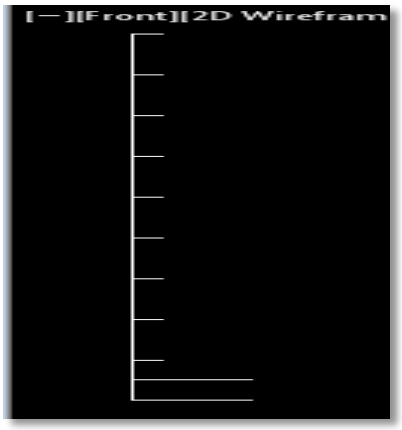
Step-2: Draw a line of **2-Inches** at bottom and take its offset of **1-inch** in upward direction.



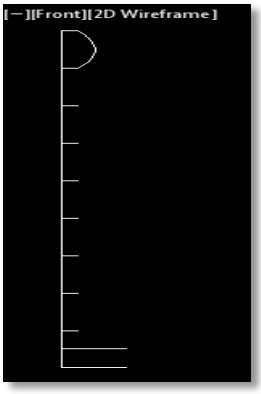
Step-3: Draw **1-inche** line from bottom to top vertically and then draw **0.5-inch** line horizontally to right direction.

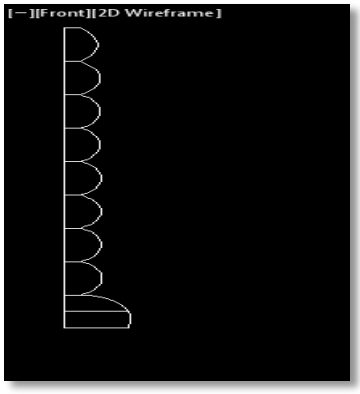


Step-4: Take Multi **offsets** of the last line through **2-inches** in upward direction

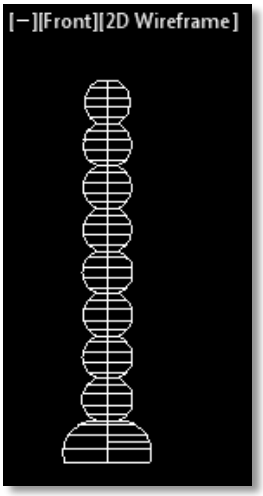


Step-5: Now use **arc** to join these lines.

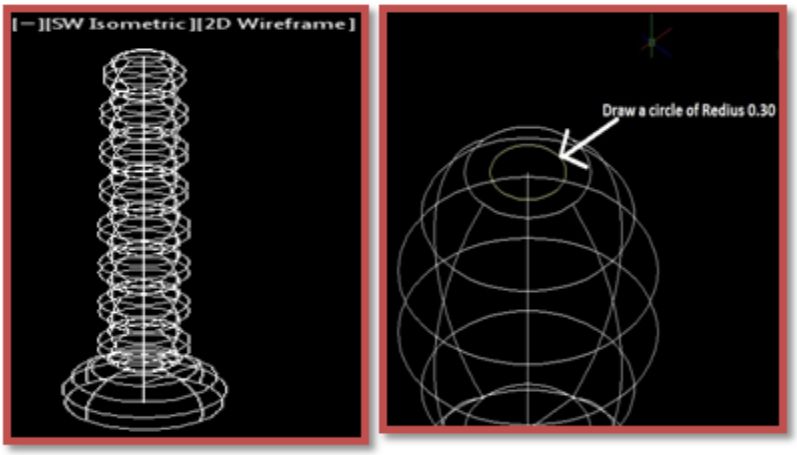




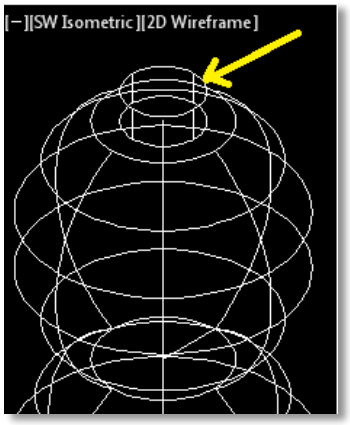
Step-6: use **revolve** command to revolve arcs in 360 angle about **the vertical line**



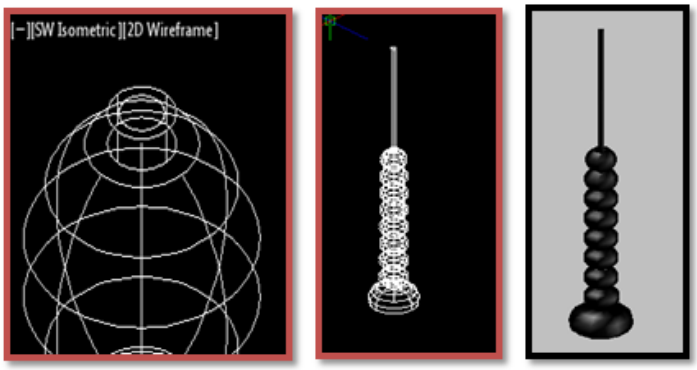
Step-7: Change the view into SW isomatric ,Zoom it and draw a circle with radius of 0.30 inches on the top as shown in figure below



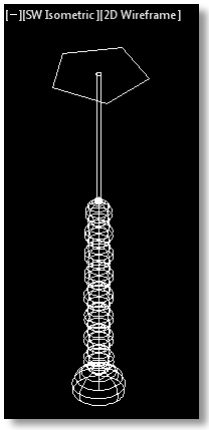
Step-8: Now extrude this circle up to 0.25 inch.



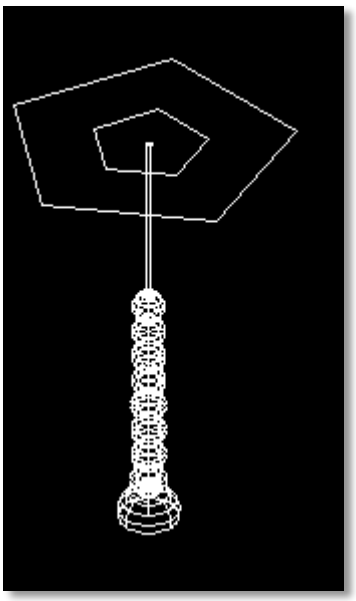
Step-9: Again draw a circle of **0.20-Inch** on the top of previously created circle and **extrude** the circle up to **12-Inches** to make a stand for the lamp.



Step-10: Draw a polygon of **5-sides** with **4-inches** radius; center point should be the top of the last created stand as shown in the figure below.

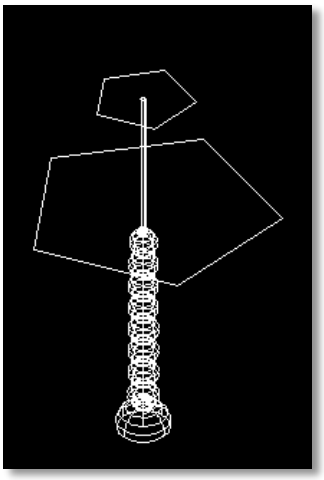


Step-11: Draw another polygon of **5-Sides** with **10-inches** radius.

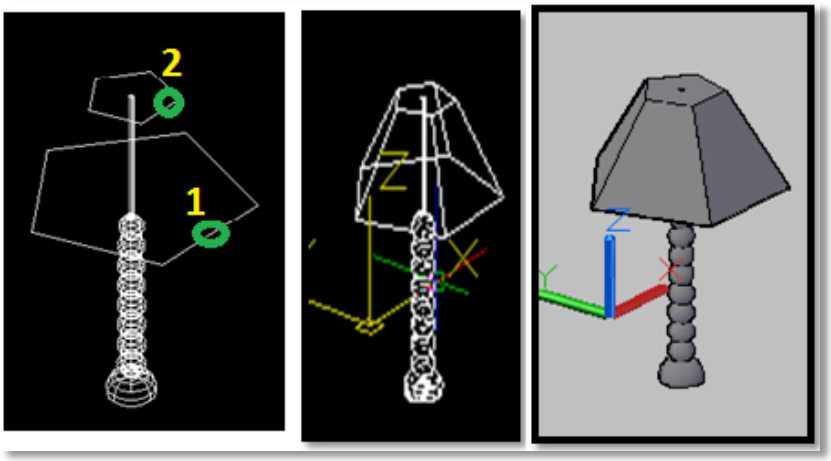


Step-12: Move the outer polygon up to 10-inches in downward direction.

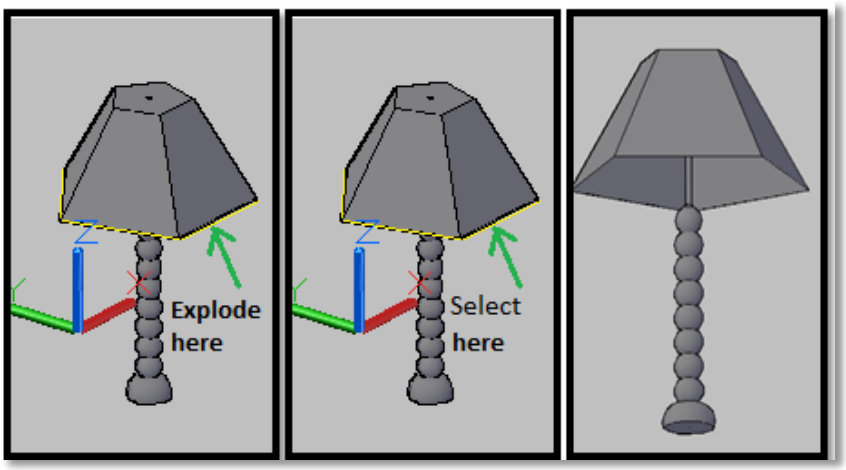
a. Type **M** command in command bar
b. Select the outer polygon
c. Move the pointer in down direction
d. Define **10-inches** in command bar and Press **Enter.**



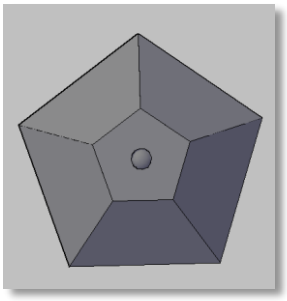
Step-13: Use loft command across these two polygons to make them solid and show it as shade.



Step-14: Use **x/xp** command to explode lines and then delete from bottom.

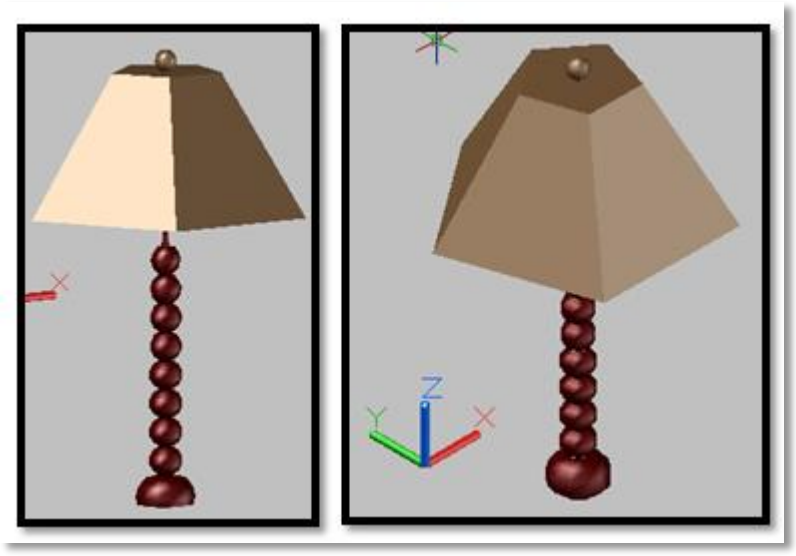


Step-15: Use sphere command in to make show a round nut on the top of the shade.



Step-16: Now apply layers/Color to stand and shade.

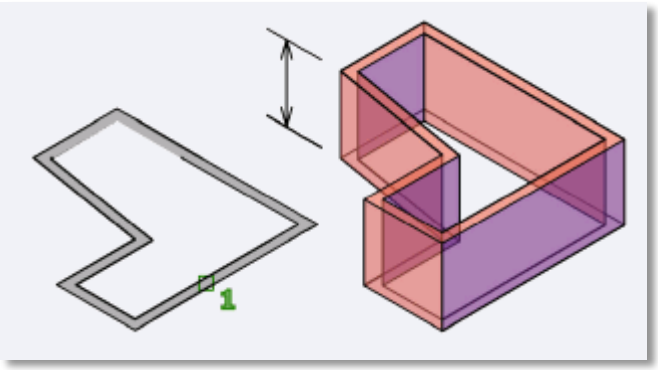
Final Look

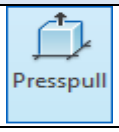


2. From Home Menu

Edit→Press Pull

Press Pull is used to extrude an object either by selecting the 2D polyline object or click within the boundary area.

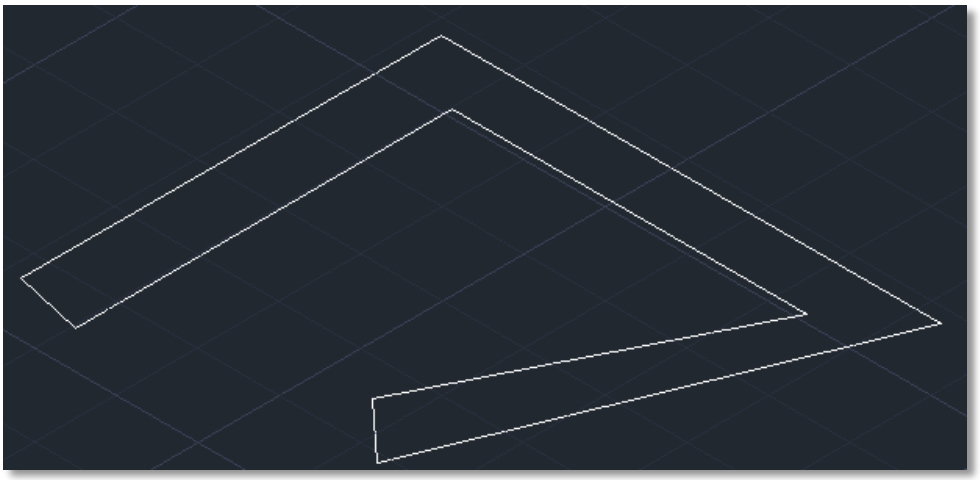


Home tab	**Button**	**Command Entry**
Press Pull		presspull

Procedure

How to use Press Pull Command

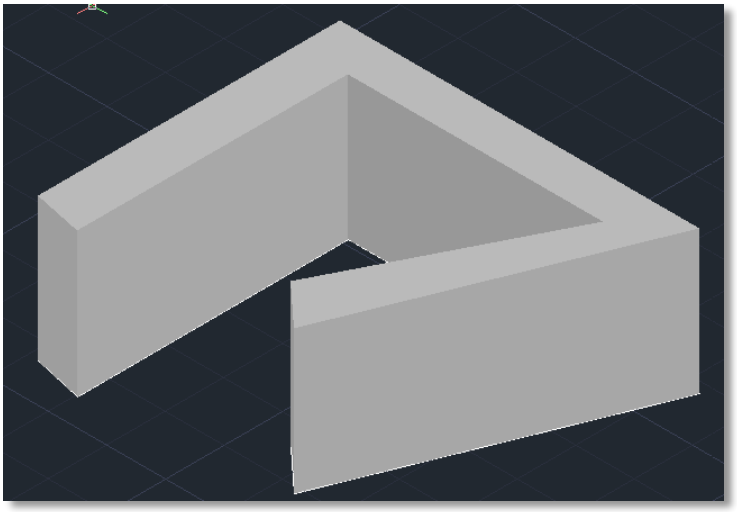
Apply press pull command to the following object



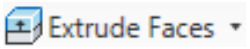
Step-1: Type Presspull Command in command bar or click create→Press Pull button.

Step-2: Select the object if the object created 2D Polyline otherwise click within the boundary area it will identify the boundary itself, then press Enter.

Step-3: Specify the height either in +ve or –ve direction, then you should get the following.



Boolean Operation: You may need to combine multiple parts into one, or remove sections from a solid. AutoCAD has some commands that make this easy for you. These are the Boolean operations as well as some other helpful commands for solids editing.

COMMAND	INPUT	ICON	DESCRIPTION
UNION (Boolean)	UNION / UNI		Joins two or more solids into creating one based on the total geometry of all.
SUBTRACT (Boolean)	SUBTRACT / SU		Subtracts one or more solids from another creating a solid based on the remaining geometry.
INTERSECT (Boolean)	INTERSECT / IN		Creates a single solid from one more solids based on the intersected geometry.
EXTRUDE FACE	SOLIDEDIT		Allows you to increase the size of a solid by extruding out one of its faces.
SLICE	SLICE		Slices a solid along a cutting plane.
3D ALIGN	3DALIGN		Aligns 3D Objects in 3D space.

Edit→Union

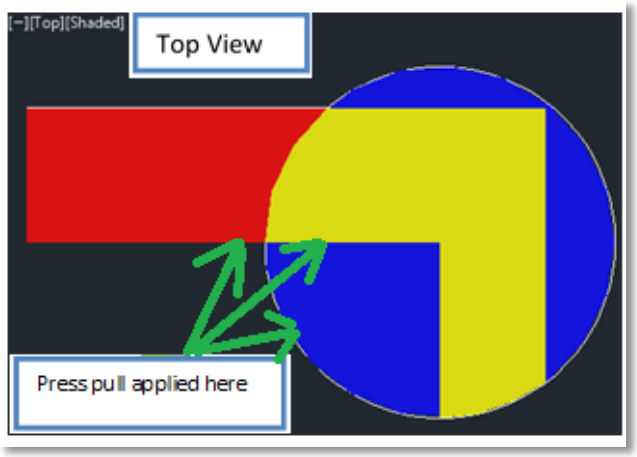
Union is used to combine two or more than two 3D objects, surfaces or 2D regions into a single object, surfaces or 2D region. The objects must be of the same type.

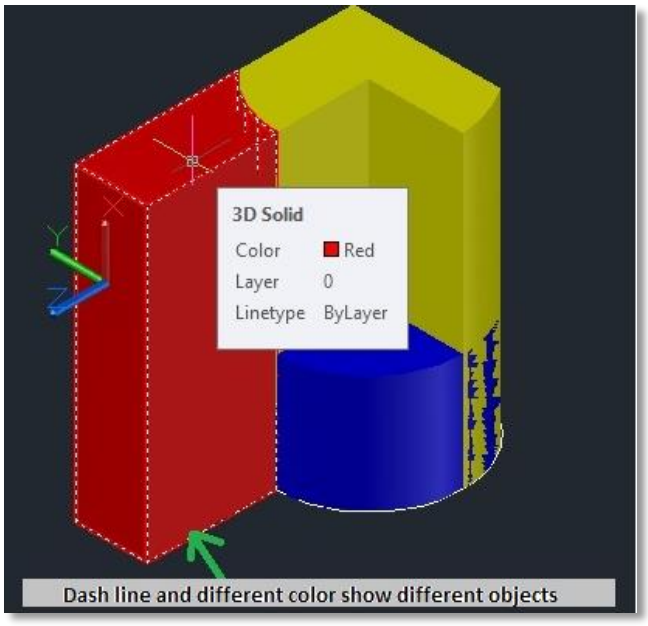
Home tab	**Button**	**Command Entry**
Union	Union	uni

Procedure

How to use union Command

Apply Union command to the objects shown in the figure below

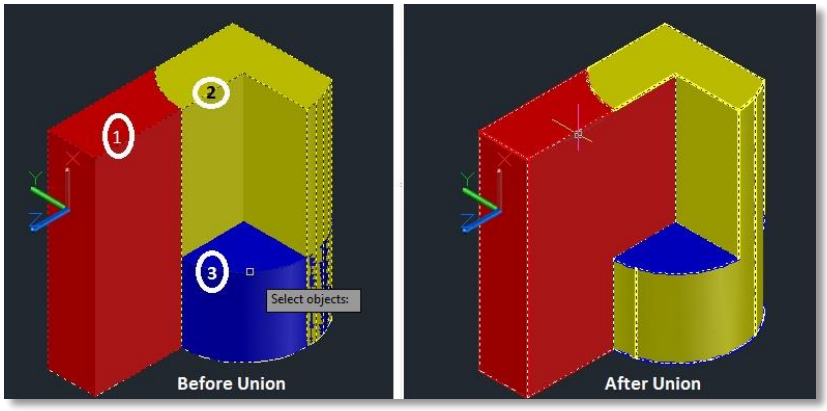




Step-1: Type uni command in command bar or click create→union button.

Step-2: select the three different objects by clicking one by one and press Enter.

Step-3: Now you will see the three selected objects as a single object.



Edit→Subtract

Subtract is used as the name define to subtract one set of object from the other object, overlapping set.

Home tab	Button	Command Entry
Subtract	Subtract	su

Procedure

How to use Subtract Command

Apply Subtract command to make door in a single wall as shown in the figure.

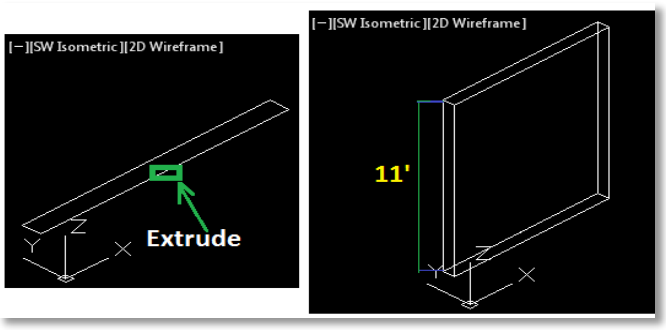
Step-1: Draw 10 feet Line horizontally shown in the figure below.

Step-2: Take offset of the line through 9 inches and make the boundary as shown in the figure below.

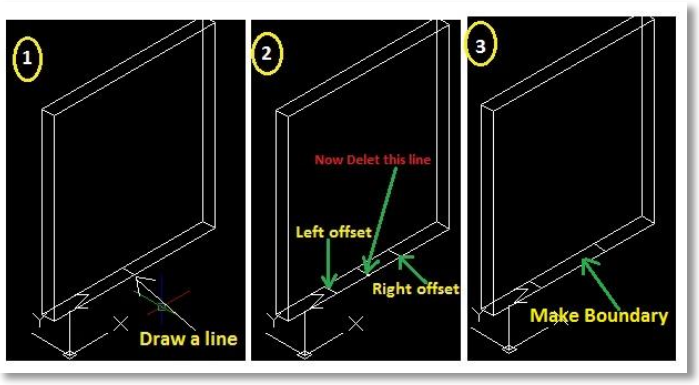
Step-3: Change the view from Top to SW Isometric and extrude the object up to 11 feet.

Do you know!

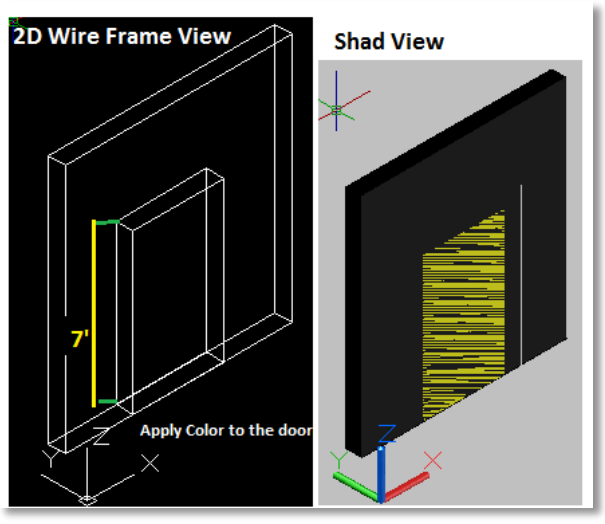
Offset: To make copy of a circle, line and curve at specified distance or through a point.



Step-4: Draw a line in between mid-points at the bottom and take offsets of 2 feet Right and left direction as shown in the figure.

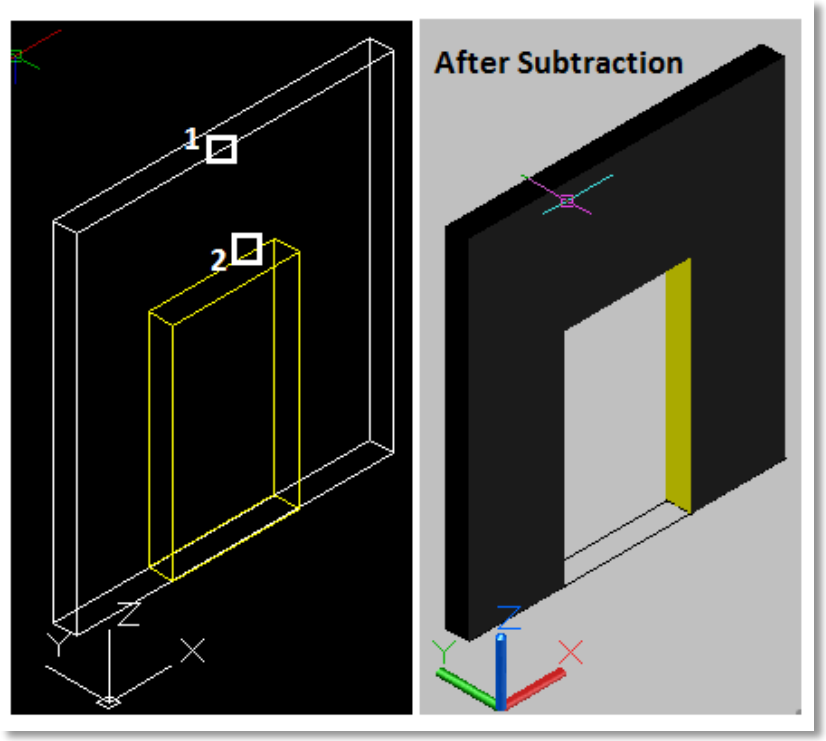


Step-5: Apply extrude command up to **7 feet** to the last created boundary in figure above.



Step-6: Now Apply Subtract Command either by put **SU** in command bar or click the subtract button.

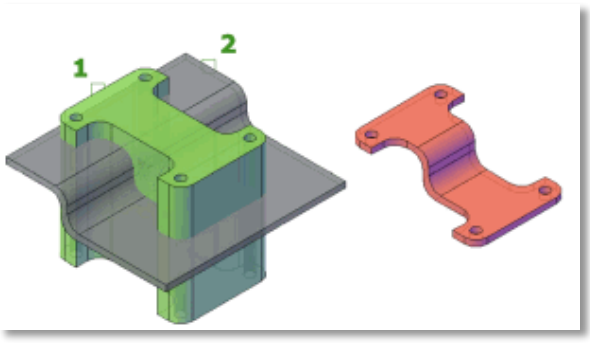
Select the object 1 that you want to keep, press **Enter**, and then select the object 2 that you want to subtract.

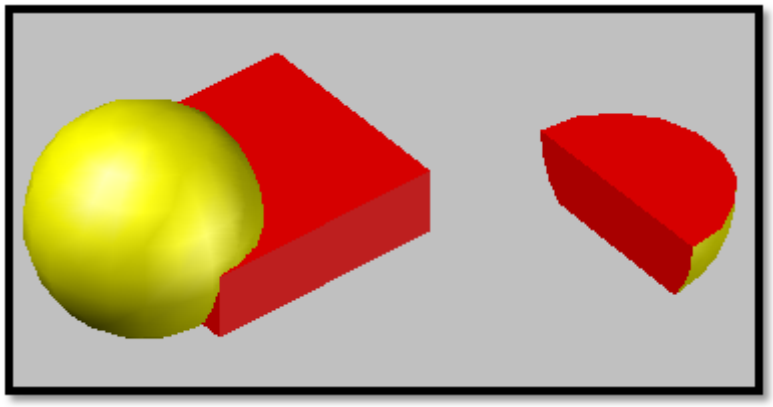


Edit→Intersect

With INTERSECT, you can create a 3D solid from the common volume of two or more existing 3D solids, surfaces, or regions.

Home tab	**Button**	**Command Entry**
Intersection	Intersect	in



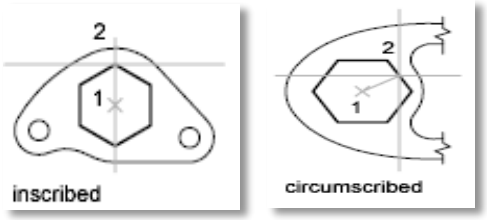


3. From Home Menu

Draw tool bar is already discussed in 2nd Module, but for knowledge enhancement we will slightly discuss the polygon here.

Draw→ Polygons

Polygons can be used to draw equilateral triangles, squares, pentagons, hexagons, and other multi-sided figures.



Home tab	**Button**	**Command Entry**
Polygon		Pol

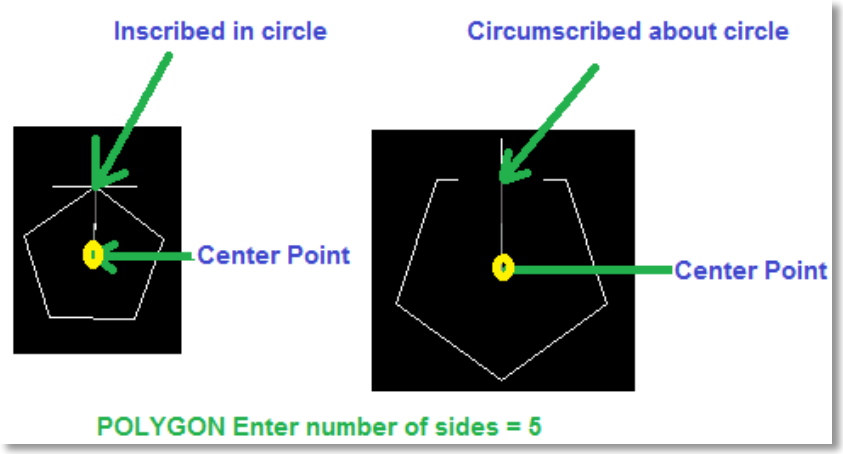
Step-1: Type **pol** command in command line or Click Home tab Draw panel **Polygon** button.

Step-2: At the Command prompt, enter the number of sides.

Step-3: Specify the center of the polygon (any point).

Step-4: Enter **c** to specify a polygon circumscribed about a circle.

Step-5: Enter the radius length.



4. From Home Menu

Layer

Layers can be very useful as you progress and make changes to a drawing in AutoCAD.

Properties of the Layer:

Status

Indicates the type of item: layer filter, layer in use, empty layer, or current layer.

Name

Displays the name of the layer or filter. Press F2 to enter a new name.

On

Turns the selected layers on and off. When a layer is on, it is visible and available for plotting. When a layer is off, it is invisible and not plotted, even if Plot is on.

Freeze

Freezes the selected layers in all viewports, including the Model tab. You can freeze layers to speed up ZOOM, PAN, and many other operations; improve object selection performance; and reduce regeneration time for complex drawings.

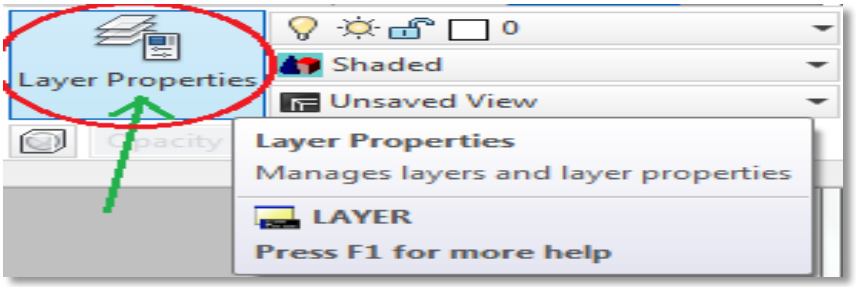
Objects on frozen layers are not displayed, plotted, hidden, or regenerated.

Procedure

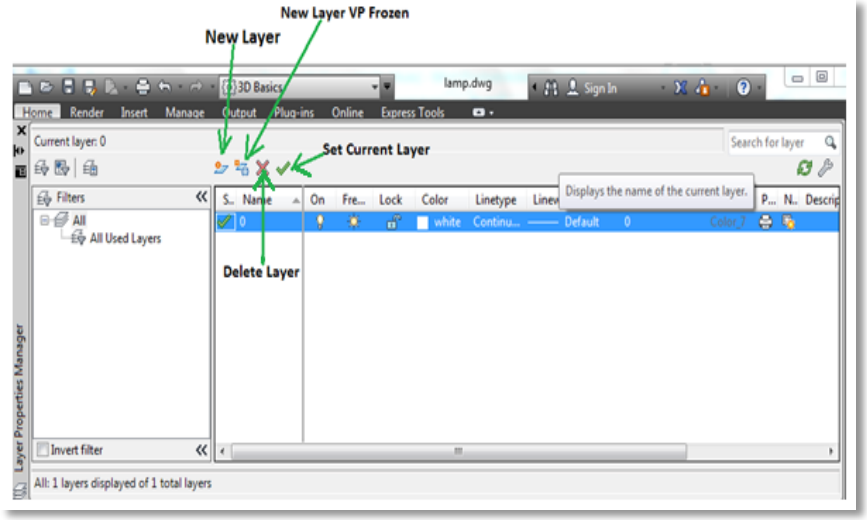
How to Create a New Layer in AutoCAD

Step-1: Type LA at the command line and then press Enter or Click the Layer Properties button in the Layers panel on the Home tab of the Ribbon.

The Layer Properties Manager palette appears. A new drawing has only one layer: Layer 0 (zero). You need to add the layers necessary for the drawing.



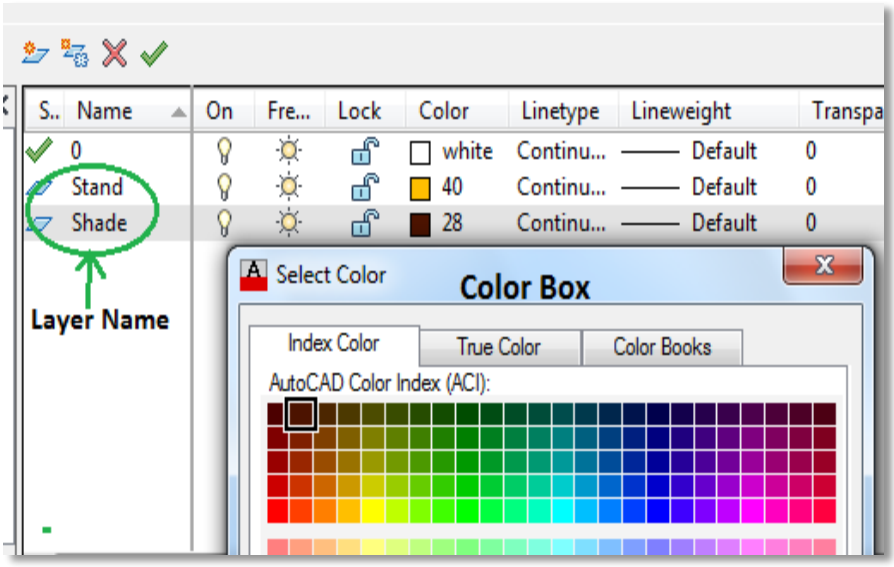
Step-2: Click the New Layer button: A new layer appears. AutoCAD names it Layer1 but highlights the name in an edit box so that you can type a new name to replace it easily.



Step- 3: Type a name for the new layer.

Step-4: On the same line as the new layer, click the color block or color name (by default, the same as the current layer) of the new layer.

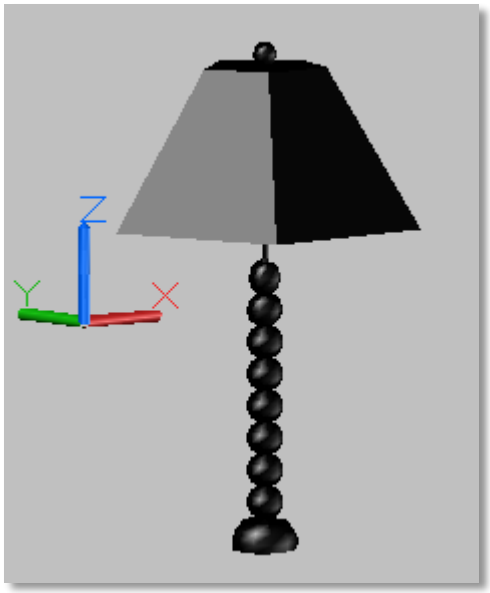
You can select any color from the color dialog box appears.



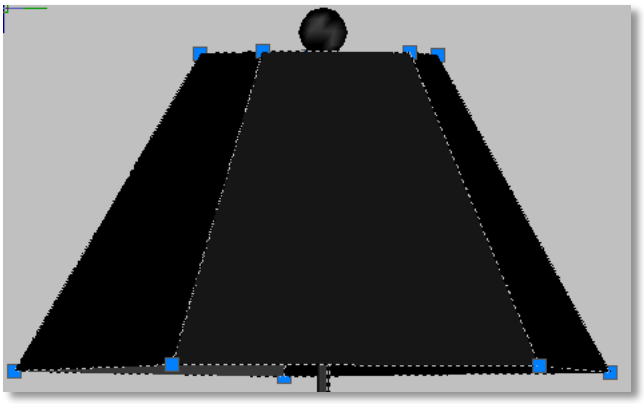
Step-5: you can select line type and line weight from the same line for the layer.

How to Apply the Layer Properties to the Object

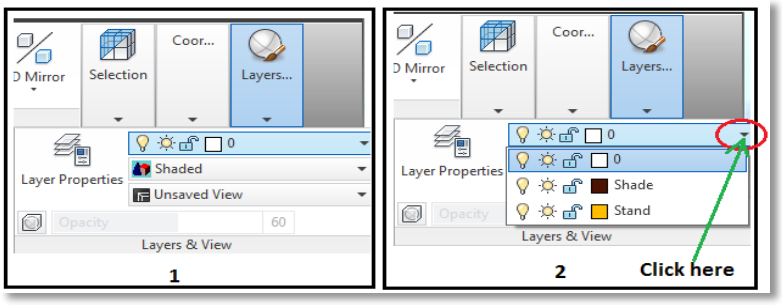
We are working with the following object to apply the properties of the layers.

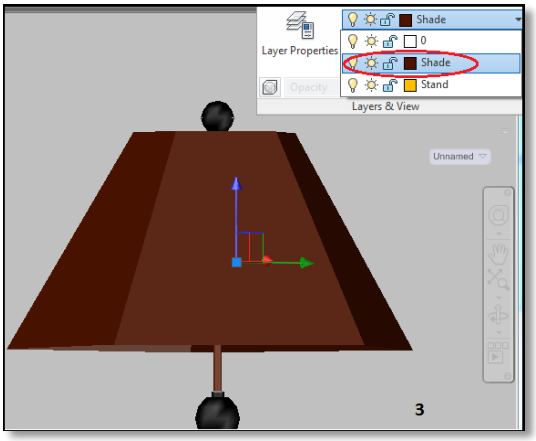


Step-1: Select the shade portion of the lamp as shown in the figure below.



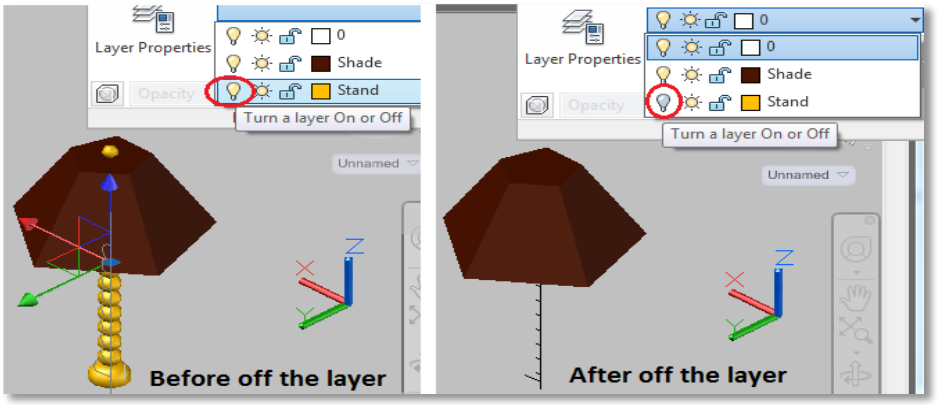
Step-2: Click the layer option from the home menu and click the shade layer as shown in the figure below





Step-3: Repeat the steps above for the stand as well, finally you should get the following

Step-4: To turn on/off the layer: Select the object and click the ON/OFF icon as shown in the figure below.



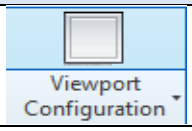
Learner's Activity

- Repeat the above steps to apply all the freeze and lock properties one by one
- Lock the layer and try to delete the object/s
- Off the layer and try to delete the object/s
- Write down the difference between the layer off Property and layer freeze Property.

6. Viewports

In model space, you can split the drawing area into one or more adjacent rectangular views known as *model space viewports*.

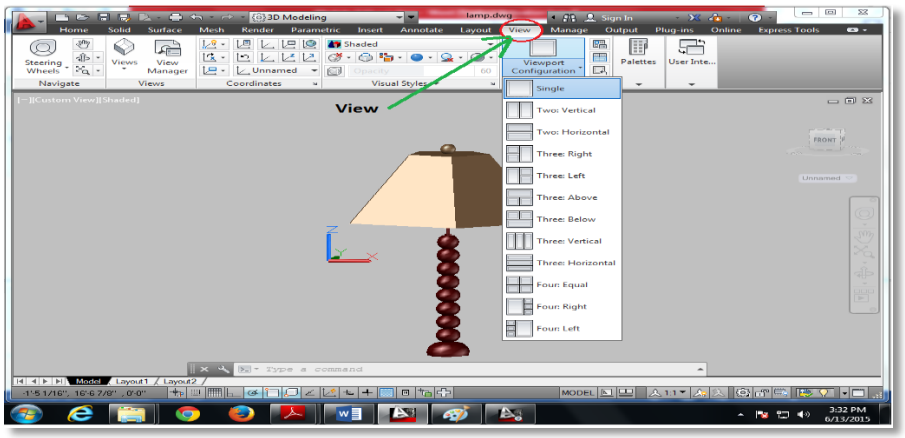
Viewports are areas that display different views of your model. In large or complex drawings, displaying different views reduces the time needed to zoom or pan in a single view. Also, errors you might miss in one view may be apparent in the others.

View tab	Button	Command Entry
Viewports		VPORTS

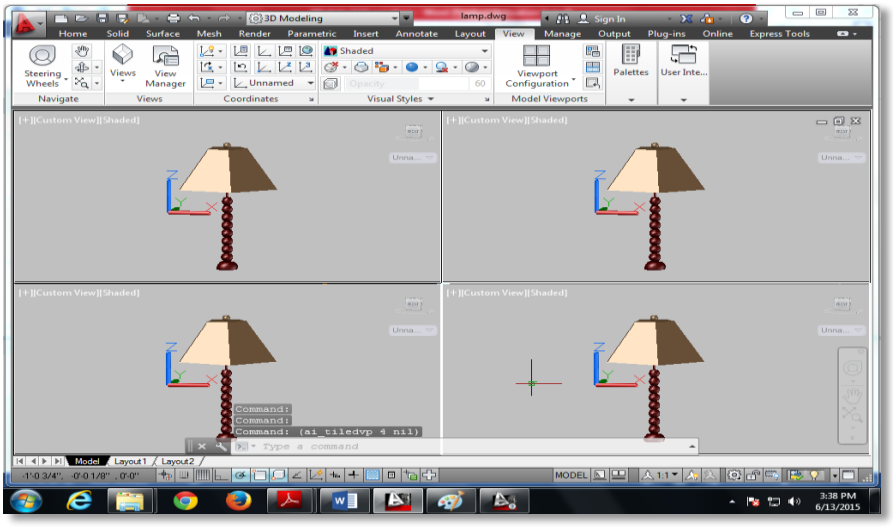
Procedure

How to Set Viewports

Step-1: Click the Viewports button on the view tab of the Ribbon.



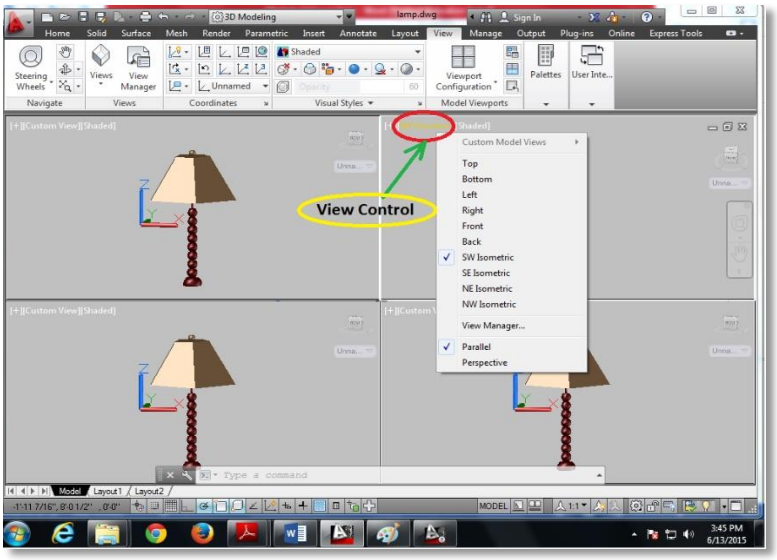
Step-2: Now select the option to split the drawing area, i.e **Four Equal** as shown in the figure below.

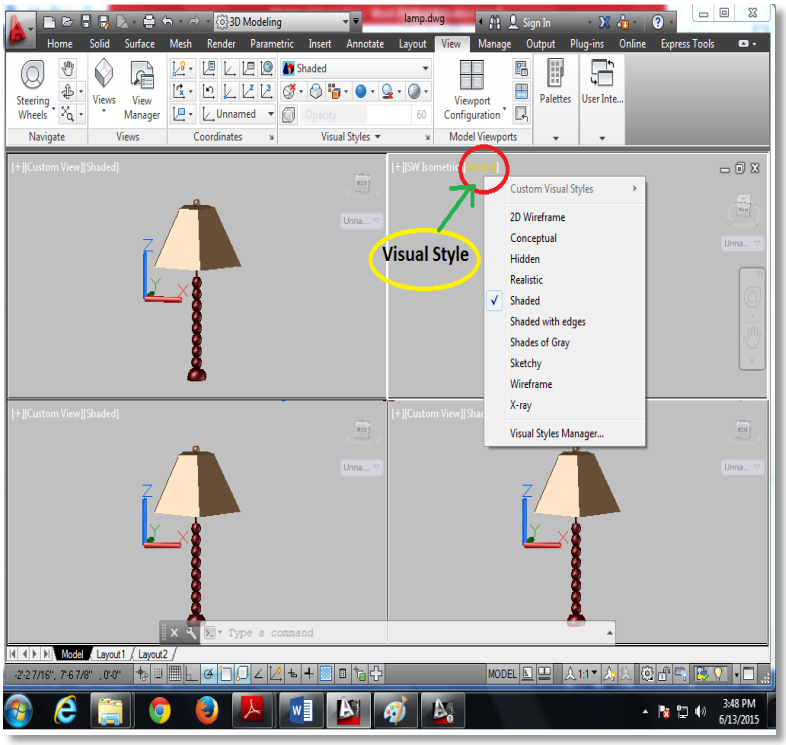


Step-3: You can work with the drawing in different view, change the views of each windows to different views and visual styles.

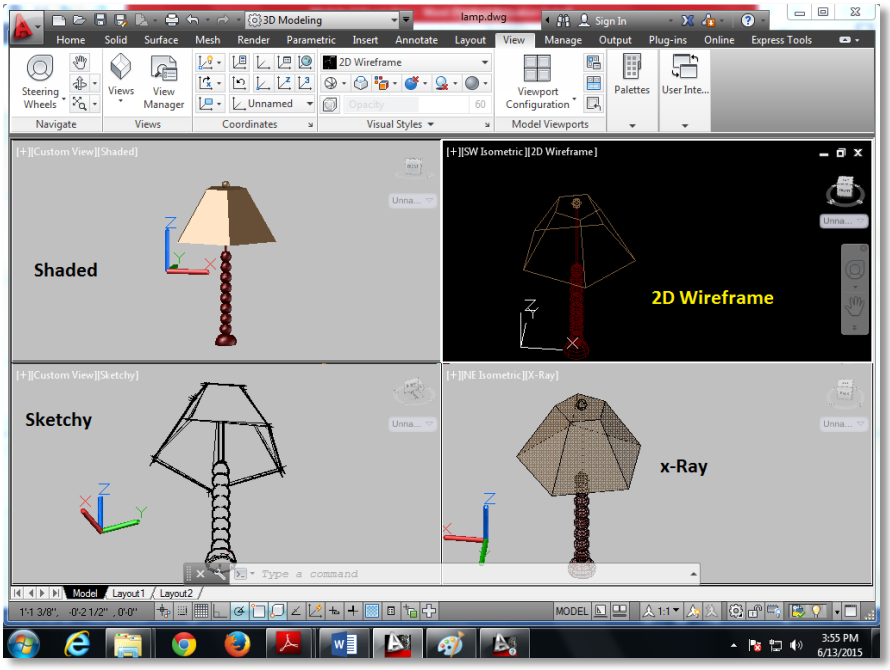
a) Just click one window and change the view and visual style

b) Repeat and change the views and visual style for all then you will get the following.





Finally you will get the following four views splitting windows



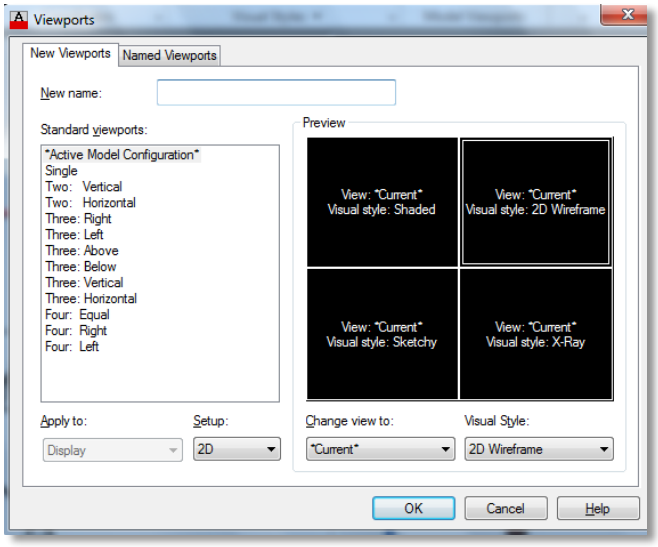
Alternate Method

7. Preset 3D Viewports

You can used Command to split the windows: by typing the **VPORTS** command in command line.

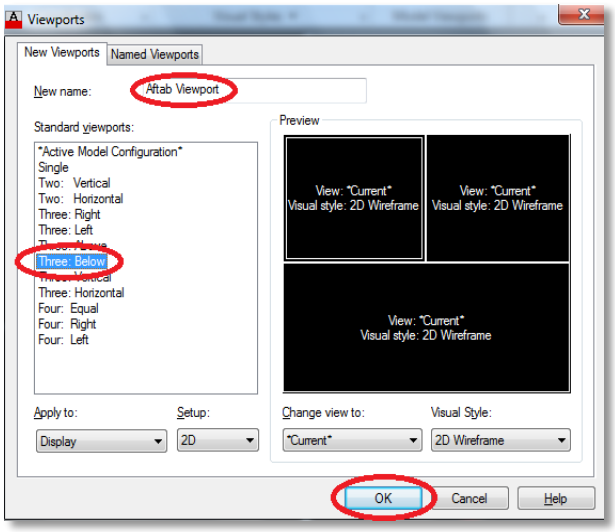
Step-1 Type **vports**

You will get the following figure.

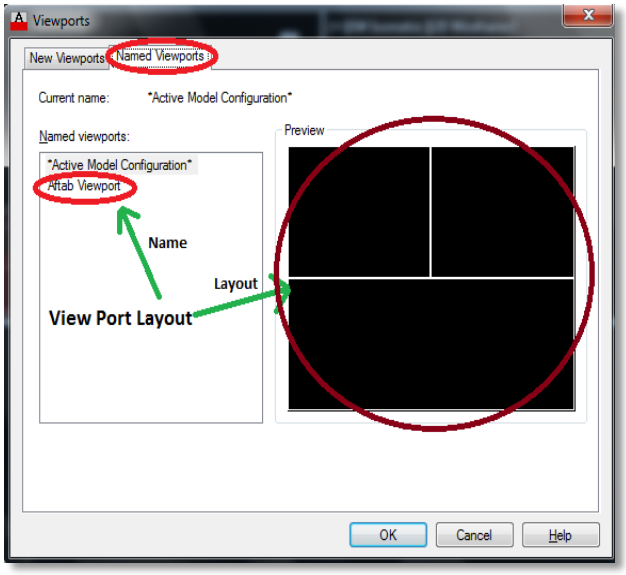


Step-2: type the name for the new model space viewport.

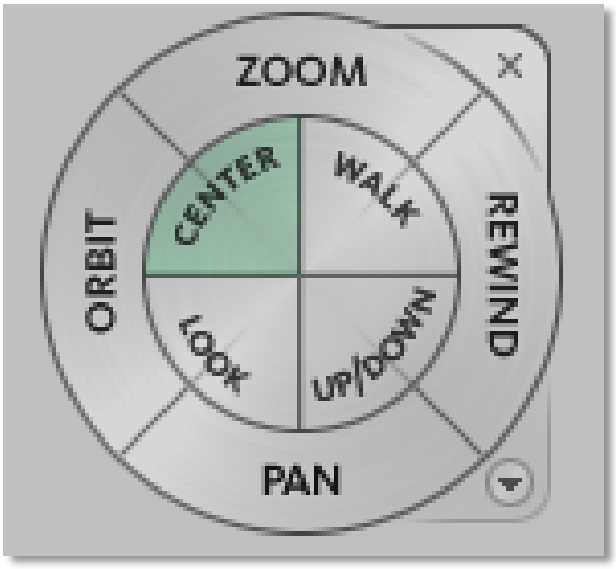
- If you do not enter a name, the viewport configuration is applied but not saved.
- If a viewport configuration is not saved, it cannot be used in a layout.
- If you specify a name it can be save and can be used in a layout.



Step-3: Whenever you used the viewport again you will get the layout named **Aftab Viewport** as shown in figure below.



8. Steering Wheel

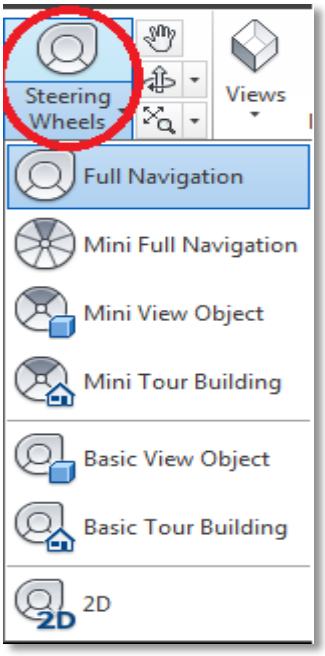


The steering wheel is a navigation tool that allows you to zoom and Pan from a single interface. It also lets you rewind through previous displays. This wheel is an all-in-one navigation tool for the both 2D and 3D use and comes in several configurations.

Steering Wheels, also known as wheels, you can save your time by combining many of the common navigation tools into a single interface. Wheels are task specific from which you can navigate and orient a model in different views.

There are different wheels available in the navigation wheel.

Click the Steering wheel on the view tab of the Ribbon.

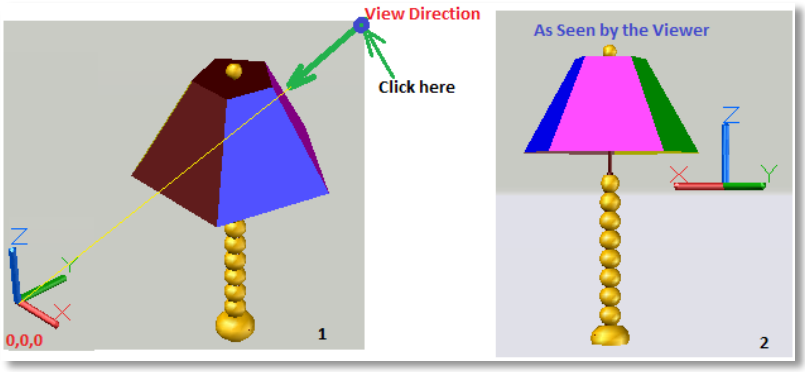


9. VPOINT

Sets the viewing direction for a 3D visualization of the drawing.

VPoint creates a vector that defines a direction from which the drawing can be viewed. The view defined is as if the viewer is looking from the point back at the origin (0, 0, 0).

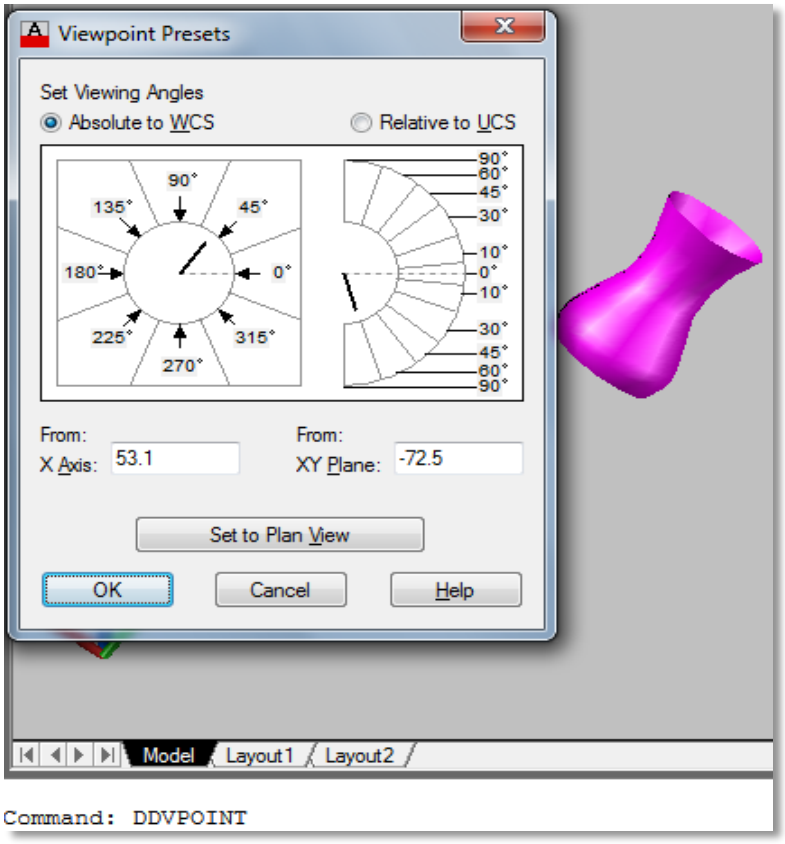
View tab	Button	Command Entry
VPOINT	AutoCAD Classic →View→3D Views→Viewpoint	VPOINT



10. DDPOINT

It is used to set a 3D viewing direction, if the standard views are not sufficient for your needs, DDVPOINT can give you both flexibility and precision.

Simple type DDVPOINT in the command bar to open the presets dialog box as shown in the figure below.


Viewpoint Presets
Set Viewing Angles
Absolute to WCS
Relative to UCS
90°
135°
45°
180°
0°
225°
315°
270°
90°
60°
45°
30°
10°
0°
10°
30°
45°
60°
90°
From:
X Axis:
53.1
From:
XY Plane:
-72.5
Set to Plan View
OK
Cancel
Help
Model
Layout1
Layout2
Command: DDVPOINT

Learning Unit 2: Introduce Thickness and Elevation

Overview

After completing this unit you will be able to Apply the Thickness command, Execute the “Thickness” command at command prompt with different values or modify general properties of an object.

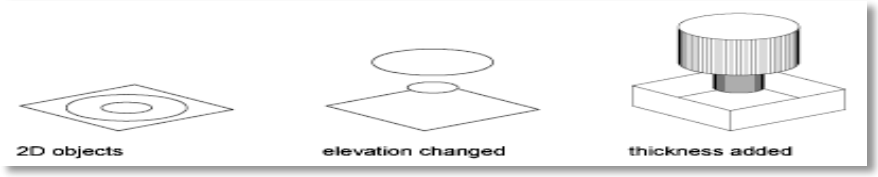
Furthermore you will be able to set the elevation property of the object from the geometry unit and Execute the “Elev” command at command.

1. Introduction to Thickness

Use the thickness property to give objects a 3D appearance.

The 3D thickness of an object is the distance that the object is extended above or below its location in 3D. Positive thickness extends upward in the positive ***Z*** direction; negative thickness extends downward (negative *Z*).

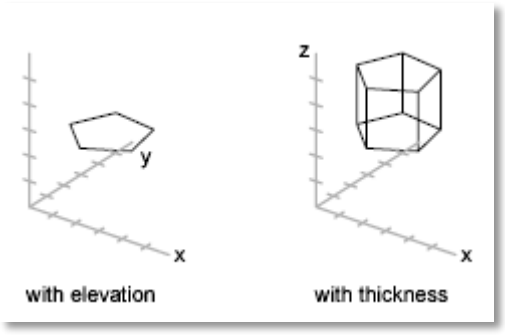
Property	**Command Entry**
Object→Property→General→Thickness	thickness



2. Elevation

The Elevation is used to sets the default *Z* value for new objects above or below the *XY* plane of the current UCS. This value is stored in the ELEVATION system variable.

Property	**Command Entry**
Object→Property→Geometry→Elevation	elev



Procedure-1

Thickness and Elevation

Step-1: Draw a rectangle

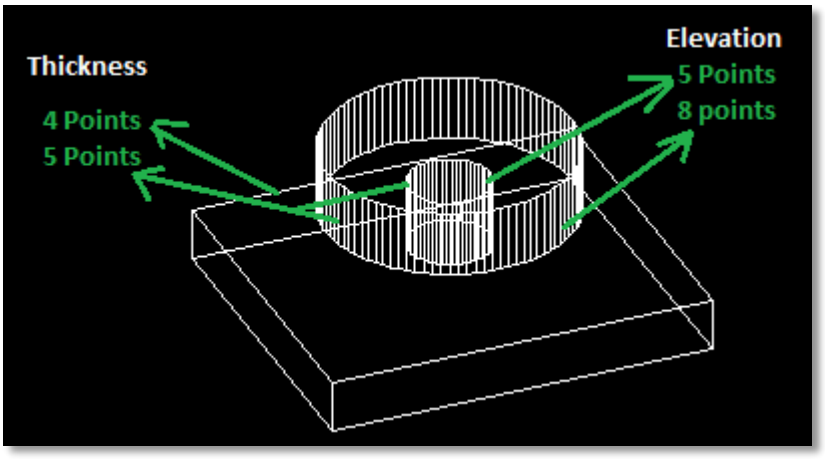
Step-2: Draw two circle (Inner and outer) within the rectangle

Step-3 elevate inner circle by **5 Points** and outer circle by **8 points**

- Select the inner circle →object property→geometry→elevation = **5**
- Select the outer circle →object property→geometry→elevation = **8**

Step-4: Apply Thickness to rectangle, inner and outer circle.

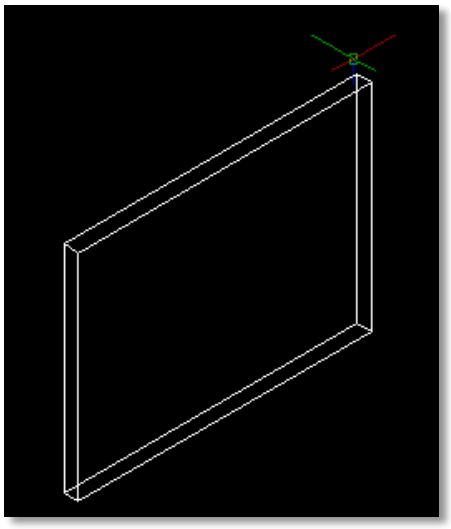
- Select the inner and outer circle →object property→General→ thickness = 5
- Select the rectangle →object property→General→thickness = **4**



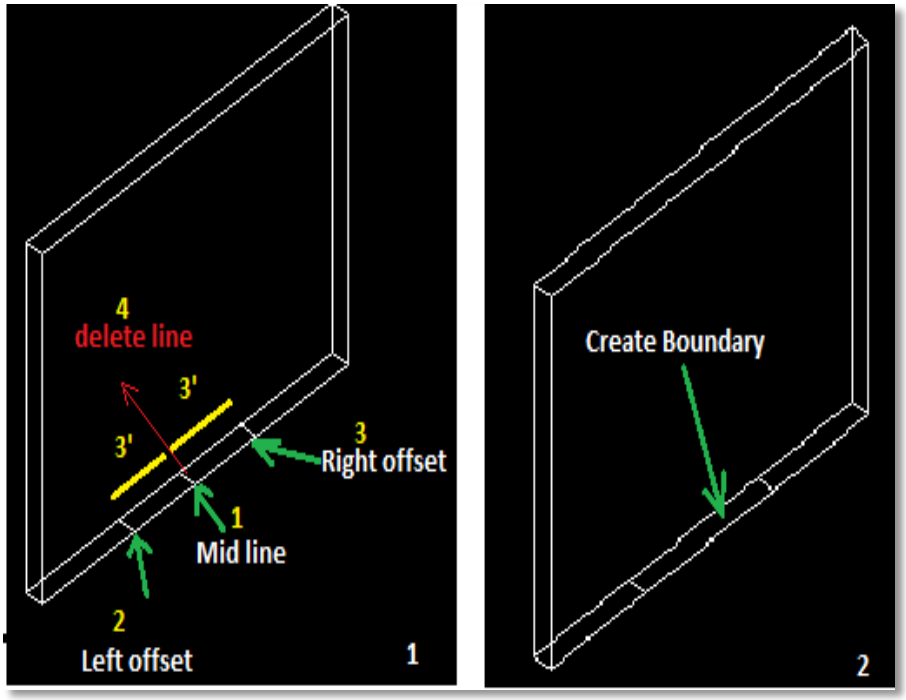
Procedure-2

Create window in a wall

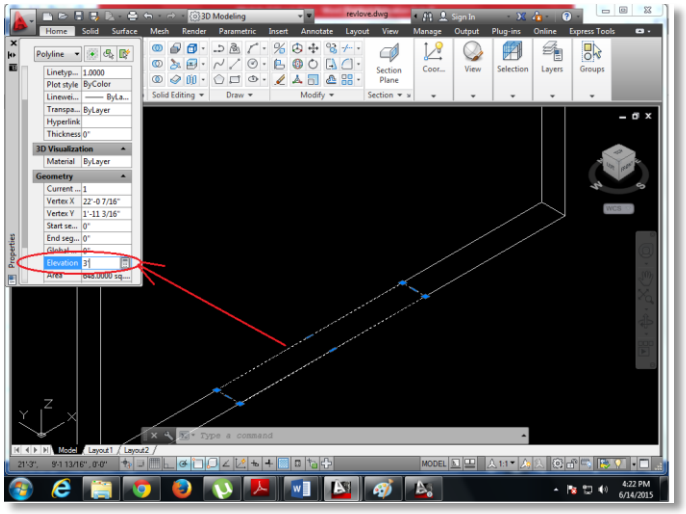
Step-1: Draw a wall with 15 feet length, 11 feet height and 9 inches of width.

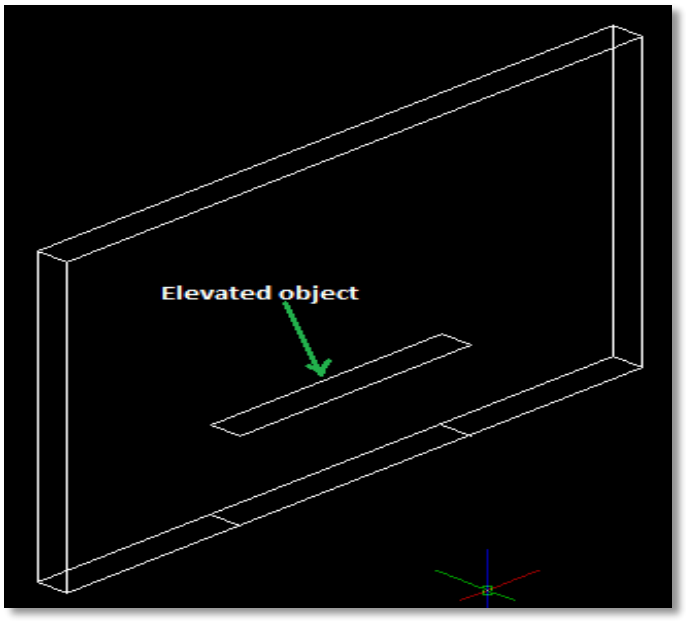


Step-2: Draw a line between mid-points in the bottom of the wall, take offsets of mid line through **3 feet**, left and right direction as shown in the figure

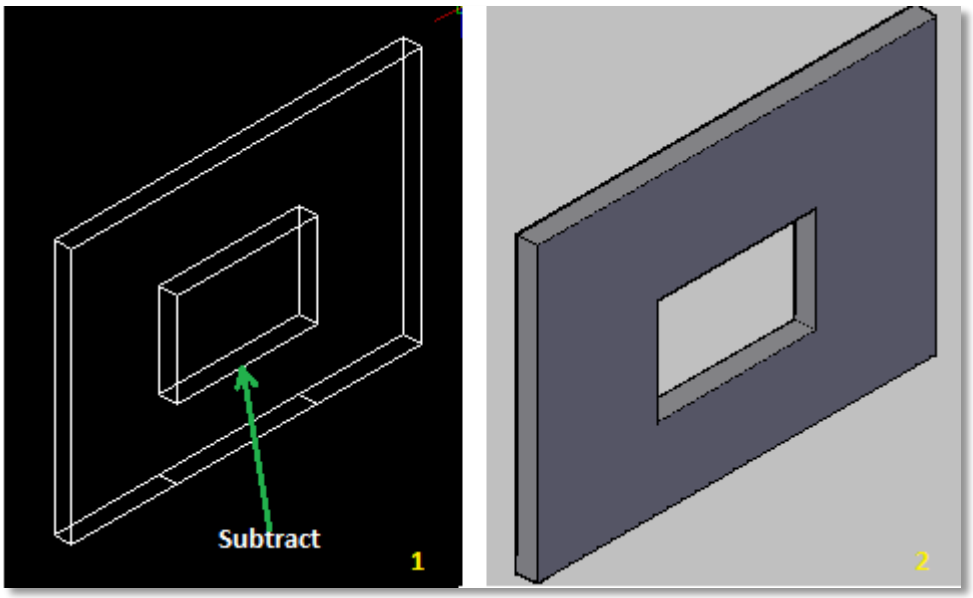


Step-3: Elevate the boundary created object by **3 feet.**





Step-4: Extrude the elevated object by 4 feet and then subtract it from the wall as shown in the figure below.


Subtract
1
2

Learning Unit 3: Visualize the Model

Overview

Note: Most of the learning elements in this unit were presented in unit-1 of this module, if you need a refresher, glance back at unit-1.

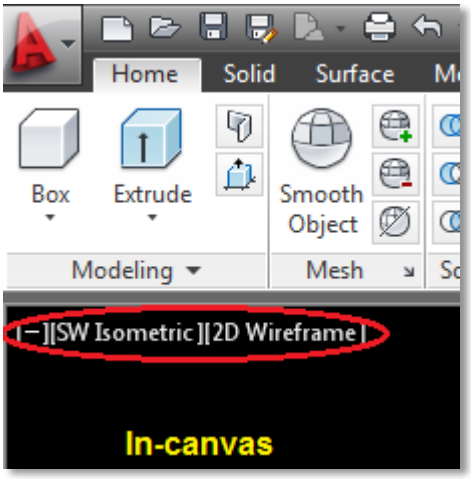
After completing this unit you will be able to Manage Visual Panel, Styles and Manager and will be able to apply different visual functions (hide, grid).

Moreover you will be able to Control the display of edges and shading (Visual Styles) in the viewport that are 2D Wireframe, 3D Wireframe, 3D Hidden, Realistic, Shaded, Shaded with Edges, Shades of Gary, Sketchy and X-Ray

1. Visual Style

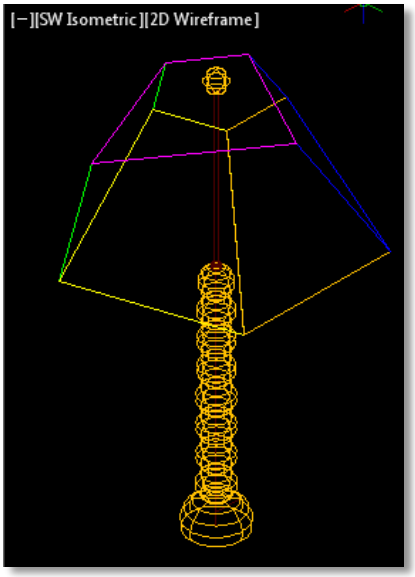
A visual style is a collection of settings that control the display of edges and shading in the viewport. Instead of using commands and setting system variables, you change the properties of the visual style. As soon as you apply a visual style or change its settings, you can see the effect in the viewport.

From the ribbon, you can change some frequently used settings or open the Visual Styles Manager or from the drawing area through In-canvas.



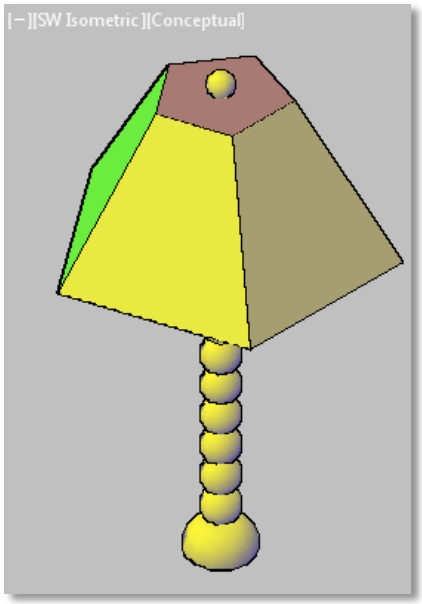
The following predefined visual styles are supplied with the product:

- **2D Wireframe.** Displays objects using lines and curves to represent the boundaries.

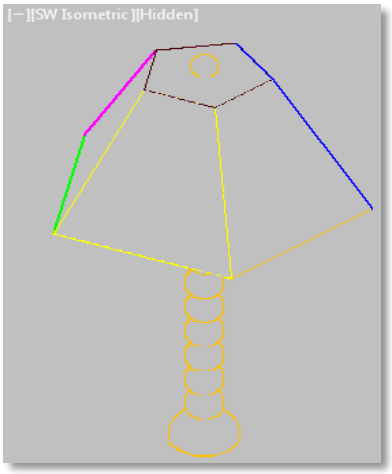


- **Conceptual.** Displays objects using smooth shading and the Gooch face style.

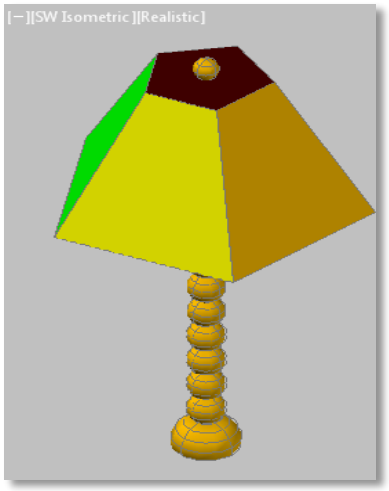
  The Gooch face style transitions between cool and warm colors, rather than dark and light. The effect is less realistic, but it can make the details of the model easier to see.



- **Hidden.** Displays objects using wireframe representation and hides lines representing back faces.



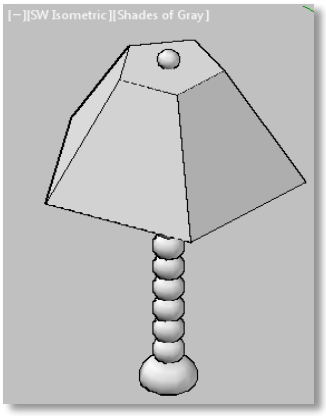
- **Realistic.** Displays objects using smooth shading and materials.



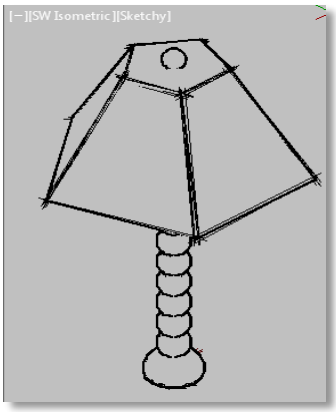
- **Shaded.** Displays objects using smooth shading.

- **Shaded with Edges.** Displays objects using smooth shading and visible edges.

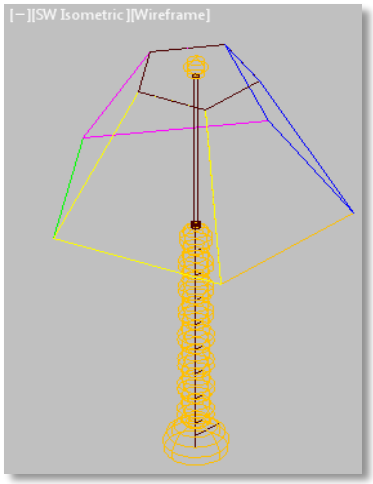
- **Shades of Gray.** Displays objects using smooth shading and monochromatic shades of gray.



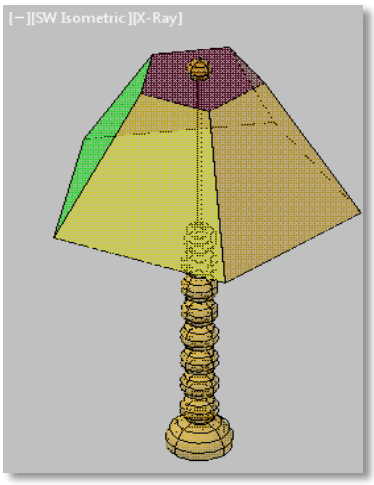
- **Sketchy.** Displays objects with a hand-sketched effect by using the Line Extensions and Jitter edge modifiers.



- **Wireframe.** Displays objects using lines and curves to represent the boundaries.



- **X-ray.** Displays objects with partial transparency.



2. Visual Style Manager

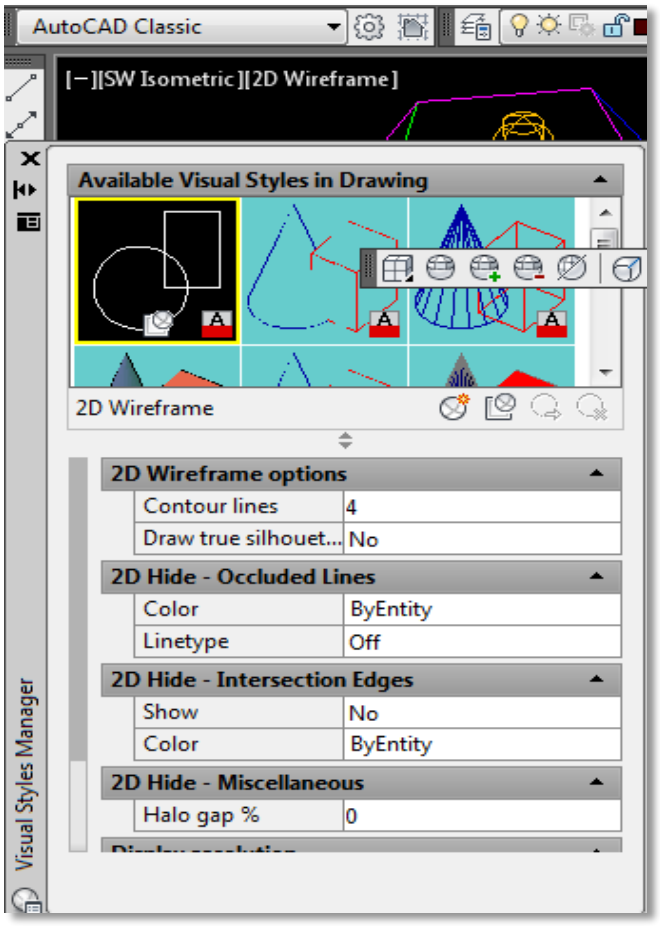
You can Creates and modifies visual styles by using visual style manager.

The Visual Styles Manager has a panel of sample images of the visual styles available in the drawing, and the following properties panels:

- Face settings
- Environment settings
- Edge settings
- Lighting settings

To view the Visual Style manager

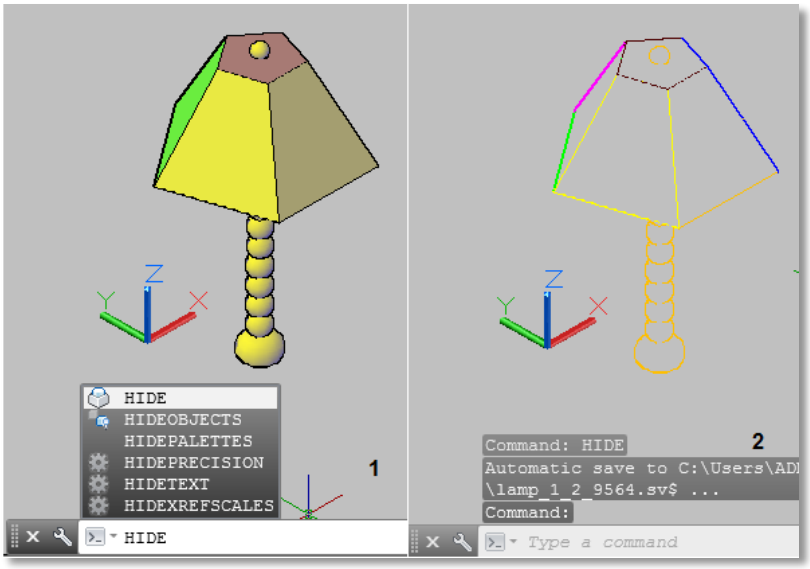
- Type **VISUALSTYLES** in command bar and Press Enter.



3. Using HIDE Command

Hide command is used to change the current view into hidden view.

- Type **hide** command in command bar and press **Enter**
  You will get the following changes



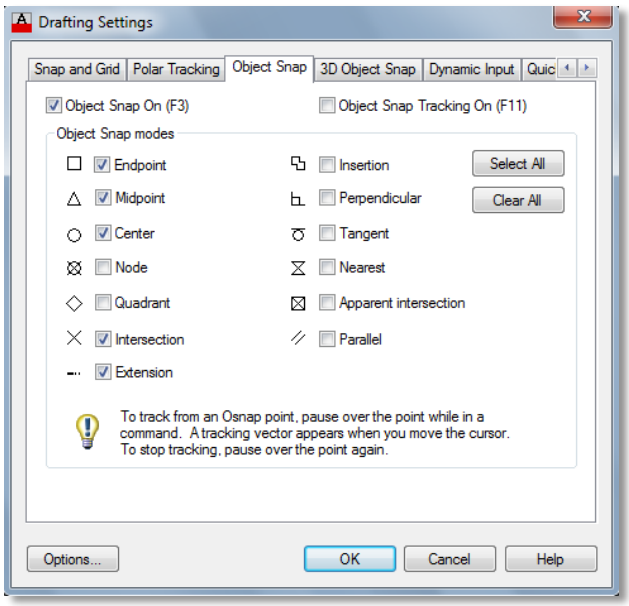
4. Using DSETTINGS Command

Note:
PLAN
Command is used in module-2, if you need a refresher, glance back to module-2.

Useful command to Sets grid and snap, polar and object snap tracking, object snap modes, Dynamic Input, and Quick Properties.

- Type **DSETTINGS** command in command bar and press **Enter**

You will get the following setting.



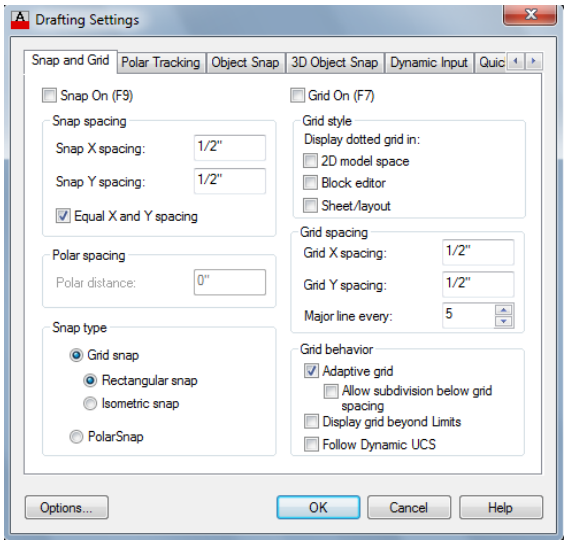
- **Alternate Method to Access**

  You can also access drafting setting by using the following method

  - **From Menu bar→**Tools→Drafting Settings

- Right-click Snap, Grid, Polar, Osnap, Otrack, Dyn or Quick Properties on the status bar. Click Settings.

To Control Snap and Grid Tab (Drafting Settings Dialog Box)



Note: The 3D Object Snap tab is not available in AutoCAD LT.

Snap On

Turns Snap mode on or off. You can also turn Snap mode on or off by clicking Snap on the status bar,

- By pressing F9, or
- By using the SNAPMODE system variable.

Snap Spacing

Controls an invisible, rectangular grid of snap locations that restricts cursor movement to specify *X* and *Y* intervals.

Snap X Spacing

Specifies the snap spacing in the *X* direction. The value must be a positive real number. (SNAPUNIT system variable)

Snap Y Spacing

Specifies the snap spacing in the *Y* direction. The value must be a positive real number. (SNAPUNIT system variable)

Equal X and Y Spacing

Forces the X and Y spacing to the same values for snap spacing and for grid spacing. The snap spacing intervals can be different from the grid spacing intervals.

Polar Spacing

Controls the Polar Snap™ increment distance.

Polar Distance

Sets the snap increment distance when Polar Snap is selected under Snap Type & Style. If this value is 0, the Polar Snap distance assumes the value for Snap X Spacing. The Polar Distance setting is used in conjunction with polar tracking and/or object snap tracking. If neither tracking feature is enabled, the Polar Distance setting has no effect. (POLARDIST system variable)

Snap Type

Sets the snap style and snap type.

Grid Snap

Sets the snap type to Grid. When you specify points, the cursor snaps along vertical or horizontal grid points. (SNAPTYPE system variable)

Rectangular Snap:

Sets the snap style to standard Rectangular snap mode. When the snap type is set to Grid snap and Snap mode is on, the cursor snaps to a rectangular snap grid. (SNAPSTYL system variable)

Isometric Snap

Sets the snap style to Isometric snap mode. When the snap type is set to Grid snap and Snap mode is on, the cursor snaps to an isometric snap grid. (SNAPSTYL system variable)

Polar Snap

Sets the snap type to Polar. When snap mode is on and you specify points with polar tracking turned on, the cursor snaps along polar alignment angles set on the Polar Tracking tab relative to the starting polar tracking point. (SNAPTYPE system variable).

Grid On

Turns the grid on or off. You can also turn grid mode on or off by clicking Grid on the status bar, by pressing F7, or by using the GRIDMODE system variable.

Grid Style

Sets the grid style in 2D contexts. You can also set grid style by using the GRIDSTYLE system variable.

Note: The limits of the grid are controlled by the LIMITS command and the GRIDDISPLAY system variable.

2D Model Space

Sets the grid style to dotted grid for 2D model space. (GRIDSTYLE system variable)

Block Editor

Sets the grid style to dotted grid for the Block Editor. (GRIDSTYLE system variable)

Sheet/Layout

Sets the grid style to dotted grid for sheet and layout. (GRIDSTYLE system variable)

Grid Spacing

Controls the display of a grid that helps you visualize distances.

Grid X Spacing

Specifies the grid spacing in the *X* direction. If this value is 0, the grid assumes the value set for Snap X Spacing. (GRIDUNIT system variable

Grid Y Spacing

Specifies the grid spacing in the *Y* direction. If this value is 0, the grid assumes the value set for Snap Y Spacing. (GRIDUNIT system variable)

Major Line Every

Specifies the frequency of major grid lines compared to minor grid lines. (GRIDMAJOR system variable)

Grid Behavior

Grid lines rather than grid dots are displayed under the following circumstances:

- AutoCAD: GRIDSTYLE is set to 0.

- AutoCAD LT: SHADEMODE is set to Hidden.

Controls the appearance of the grid lines.

Adaptive Grid

Limits the density of the grid when zoomed out. (GRIDDISPLAY system variable)

- Allow Subdivision Below Grid Spacing:

Generates additional, more closely spaced grid lines when zoomed in. The frequency of these grid lines is determined by the frequency of the major grid lines. (GRIDDISPLAY and GRIDMAJOR system variables)

Display Grid beyond Limits

Displays the grid beyond the area specified by the LIMITS command. (GRIDDISPLAY system variable)

Follow Dynamic UCS

Changes the grid plane to follow the XY plane of the dynamic UCS. (GRIDDISPLAY system variable)

Note: Not available in AutoCAD LT.

TIP: You can turn the adaptive grid on/off from the status bar.

Summary of Module

- In this module we acquainted with the AutoCAD 3D basic interfaces, which comfort learner to create/develop 3D Drawing by using 3D Solid, extrude, sweep, loft etc. commands and familiar to make modification in 3D objects with respect to different coordinates system in different workspaces and can easily identify 3D basic and 3D modeling ribbons which help the learner to execute the pull-down menus including new feature of AutoCAD , **"Navigation tools" and "View Cube"** that can save time by combining common tools into a single interface.

- In The module we also learnt the visualization of an object through In-Canvas including viewport controls, view controls and visual style controls which provide access to→ **"Multiple viewport configurations"**, **"Standard and custom views**, **and 3D projections**" and **"Standard and custom visual style"** respectively.

Frequently Asked Questions (FAQs)

FAQ 1: OSNAP Stand for?

Answer: Object Snap

FAQ 2: Enlist of object snap modes?

Answer:

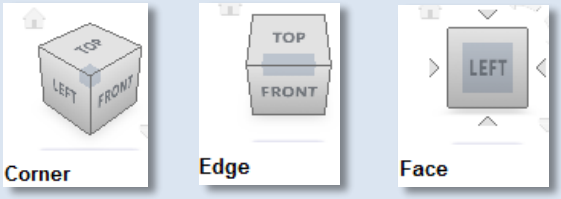
ENDpoint	CENter	TANgent
MIDpoint	NODe	NEArest
INTersection	QUAdrant	PARallel
EXTension	INSertion	
APParent Intersection	PERpendicular	

FAQ 3: What is the function of View cube?

Answer:

Reorient the current view of a model by clicking predefined areas on the ViewCube or dragging the ViewCube tool.

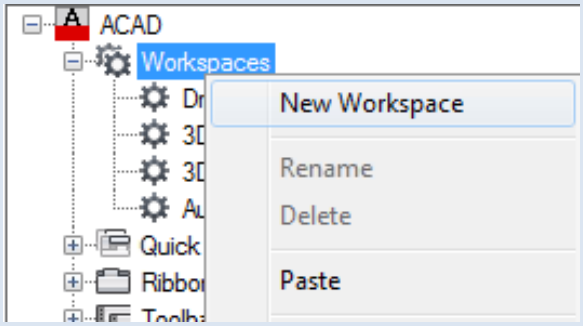
- The ViewCube provides twenty-six defined parts to click and change the current view of a model. The twenty-six defined parts are categorized into three groups: corner, edge, and face.



FAQ 4: How to Create a Workspace Using the CUI Editor?

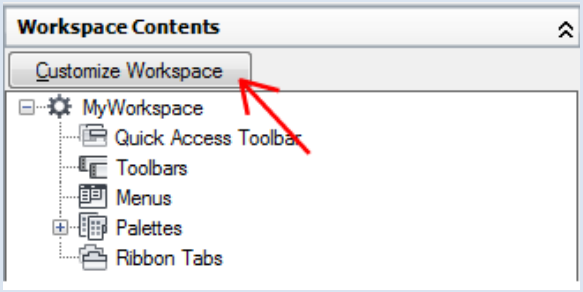
Answer:

1. In the Customize tab, in the Customizations In <file name> pane, right-click the Workspaces tree node, and select New Workspace.

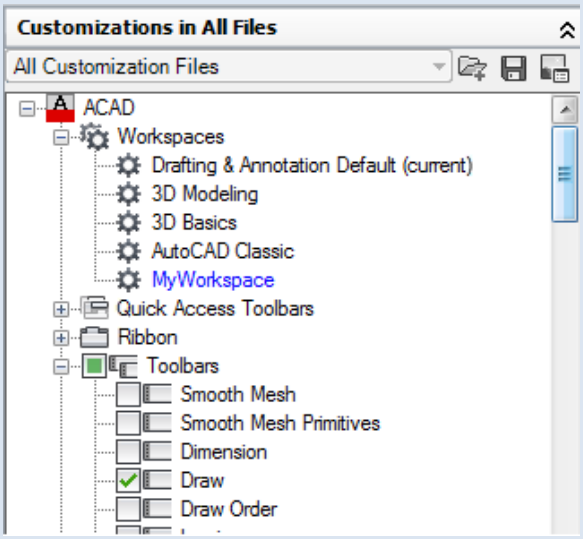


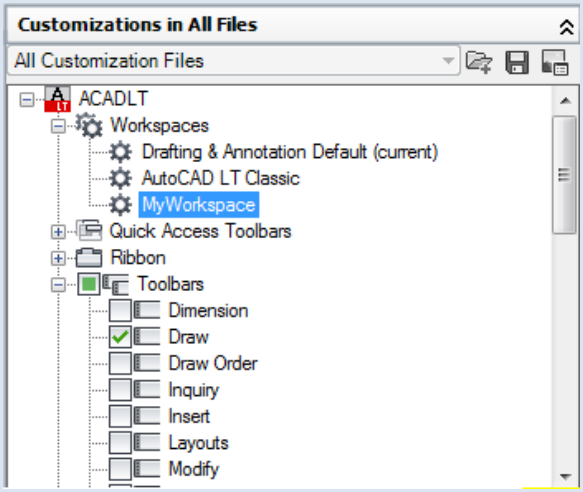
A new, empty workspace (named Workspace1) is placed at the bottom of the Workspaces tree node.

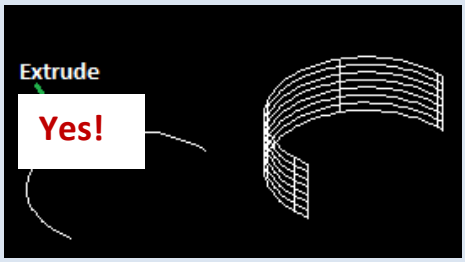
2. Do one of the following to rename the workspace:
   - Enter a new name over the default name Workspace1.
   - Right-click Workspace1. Click Rename. Then, enter a new workspace name.
   - Click Workspace1, wait, and click again over the workspace's name again to edit its name in-place.
3. In the Workspace Contents pane, click Customize Workspace.



4. In the Customizations In <file name> pane, click the plus sign (+) next to the tree nodes to expand them.
5. Click the check box next to each user interface element that you want to add to the workspace.





	The selected user interface elements are added to the workspace. 6. In the Workspace Contents pane, click Done. 7. Click Apply.
FAQ 5:	Define minimum and maximum sides of Polygon?
Answer	Minimum sides = 3 Maximum = 1024
FAQ 6:	Single line can be extruded?
Answer	
FAQ 7:	Can we switch revolve solid into revolve surface through command line?
Answer	Yes! Type rev <Enter> type MO <Enter> type su <Enter> then select Object to revolve.
FAQ 8:	Can an object be revolve in 180 degree?
Answer	Yes! Specify angle of revolution <360/270/180/90/Other> : <ENTER>
FAQ 9:	Layer can be deleted in AutoCAD?
Answer	Yes!
FAQ 10:	Which option used to change the elevation of the circle?
Answer	Good Question! Position Z instead of elevation is used to change the elevation.

Test Yourself!

Please mark the correct one from the given options. You can check your answer with the Answer Key at the end of this module.

1. Layer cannot be deleted when its property is?

a. Hide
b. Lock
c. Freeze
d. Both b and c

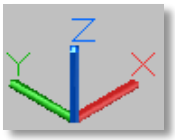
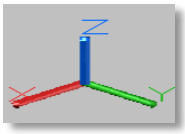
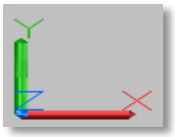
2. Short cut key for Grid on/off

a. Ctrl + g
b. F3
c. F7
d. F8

3. Short command for Setting grid and snap?

a. SNAPMODE
b. DSETTINGS
c. SNAPUNIT
d. None

4. Default Coordinated position of SW Isometric is?

a 
b 
c 
d None

5. To control the grid you must enter the ----------command?

a Grid
b Limit
c Limits
d GRIDDISPLAY

Answer Key

MCQ Number	Correct Answer
1	d
2	c
3	b
4	a
5	c

AUTOCAD

Module-4

LEARNING GUIDE

National Vocational Certificate Level 3

Version 1 - July 2013

Module 4: Draw Coordinates

Learning Outcomes

After completion of this learning module, you will be able to:

- Explain Basic terminologies of Z Coordinates
- Define User Coordinates System

Learning Unit 1: Explain Basic Terminologies of Z Coordinates

Overview

This unit describes Basic terminologies of Z Coordinates and user coordinate system, after completing this module you will be able to manipulate 3D Cartesian coordinates against **3DPOLY** command by specifying start and end points and to track the z coordinates by **osnap** option.

Further you will be able apply HILEX command to develop 3D Spiral by defining number of turns, diameter and height with the help of different functions to User coordinates systems and generating different views through View Cube

1. Cartesian Coordinates

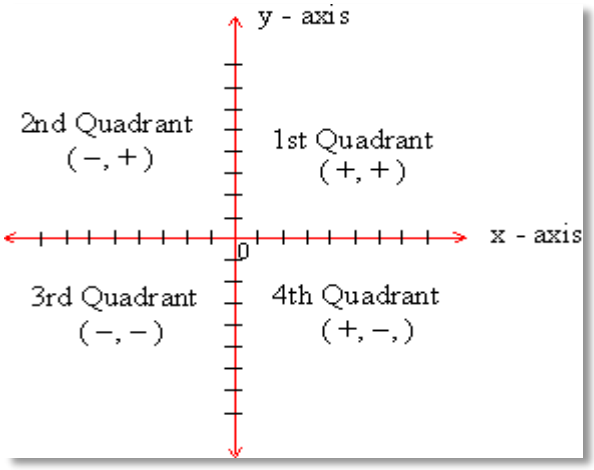
Real object can be view in 3 dimension (length, Width , Length breath and height/depth), In AutoCAD these 3 dimension is represented with three Position X,Y and Z axis, is known as the Co-ordinate System.

A *coordinate system* consists of numbered scales that identify an initial or base point and the direction for measuring next points on a graph with respect to axis.

The axes divide the coordinate system into four regions called *quadrants*. Quadrant-I is the region above the x-axis and to the right of the y-axis. Quadrant-II is the region above the x-axis and to the left of the y-axis. Quadrant-III is the region below the x-axis and to the left of the y-axis. Quadrant-IV is the region below the x-axis and to the right of the y-axis.

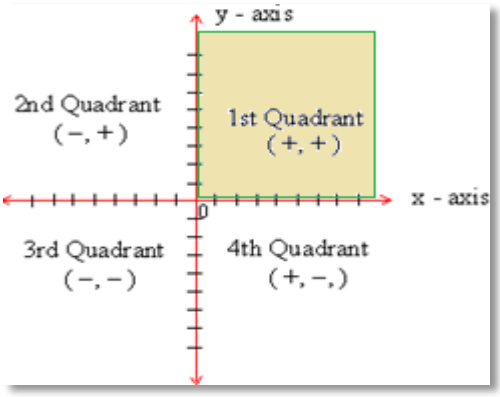
Note: The *Cartesian coordinate system*-named after the philosopher Rene Descartes, who defined the xy-coordinate system in the 1600s-consists of three numbered scales, called the x-axis, y-axis, and z-axis, that are perpendicular to each other and extend infinitely in each direction.

To use Cartesian coordinates to specify a point, enter an *X* value and a *Y* value



separated by a comma. The *X* value is the positive or negative distance, in units, along the horizontal axis. The *Y* value is the positive or negative distance, in units, along the vertical axis.

Most of your work in AutoCAD will be done in Quadrant-I, when you first open the AutoCAD drawing you will get Quadrant-I.



2D Cartesian Coordinates:

In 2D Cartesian coordinates system, you specify points on the *XY* plane, also called the *work plane*. The work plane is similar to a flat sheet of grid paper. The *X* value of a Cartesian coordinate specifies horizontal distance, and the *Y* value specifies vertical distance. The origin point (0, 0) indicates where the two axes intersect.

Enter the following command to get the following drawing:

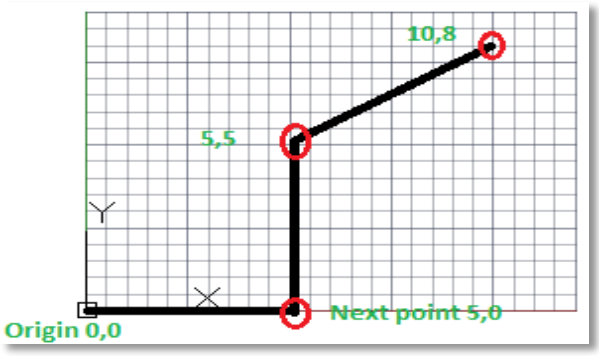
Step-1: Type L

Step-2: 0, 0 **<Enter>**

Step-3: 5, 0 **<Enter>**

Step-4: 5, 5 **<Enter>**

Step-5:10, 8**<Enter>**



Enter the following command to get the following drawing:

Step-1: Command: L

Step-2: Specify first point: 7, 2

Step-3: Specify next point or [Undo]: 11, 3

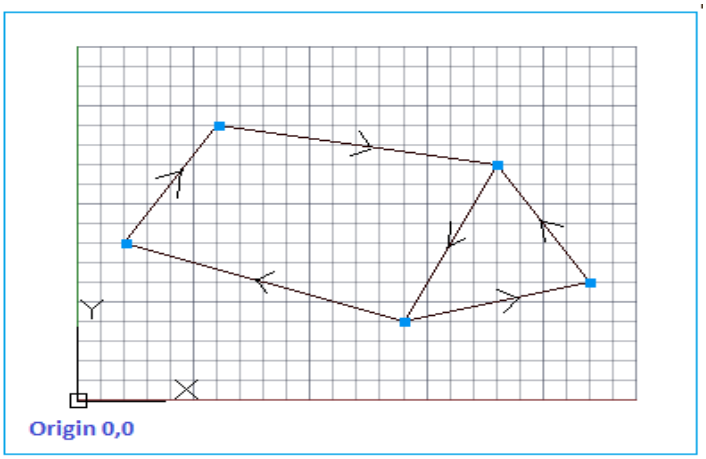
Step-4: Specify next point or [Undo]: 9, 6

Step-5: Specify next point or [Close/Undo]: 7, 2

Step-6: Specify next point or [Close/Undo]: 1, 4

Step-7: Specify next point or [Close/Undo]: 3, 7

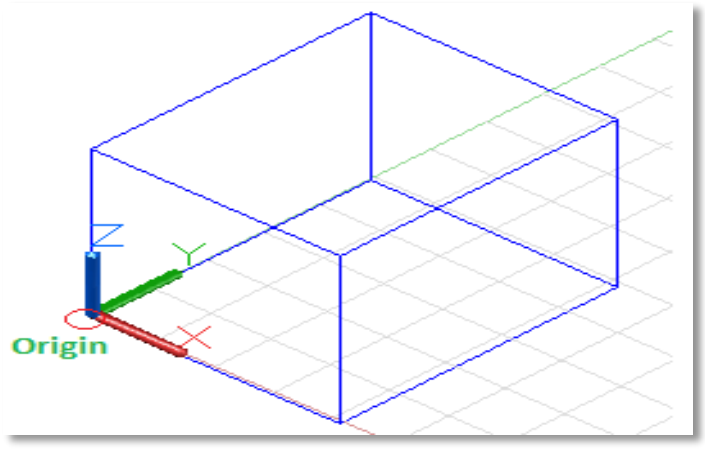
Step-8: Specify next point or [Close/Undo]: 9, 6



3D Cartesian Coordinates:

3D Cartesian coordinates specify a precise location by using three coordinate values: *X*, *Y*, and *Z*.

Entering 3D Cartesian coordinate values *(X,Y,Z)* is similar to entering 2D coordinate values *(X,Y)*. In addition to specifying *X* and *Y* values, you also specify a ***Z*** value using the following format: ***X, Y, Z.***



3DPOLY Command:

A 3D polyline is the same as 2D Polyline connected sequence of straight line segments created as a single object. 3D polylines can be non-coplanar; however, they cannot include arc segments.

The 3D Polyline command works in exactly the same way as the Polyline command. The main difference between a normal polyline and a 3D polyline is that each vertex of a 3D polyline can have a different value for Z (height). In normal (2D) polylines, all vertexes must have the same Z value.

Draw tab	Button	Command Entry
Line→3DPolyline		3dpoly

Procedure
How to draw a 3DPOLYLINE

Enter the following command to get the following drawing:

Step-1: Command: 3DPOLY

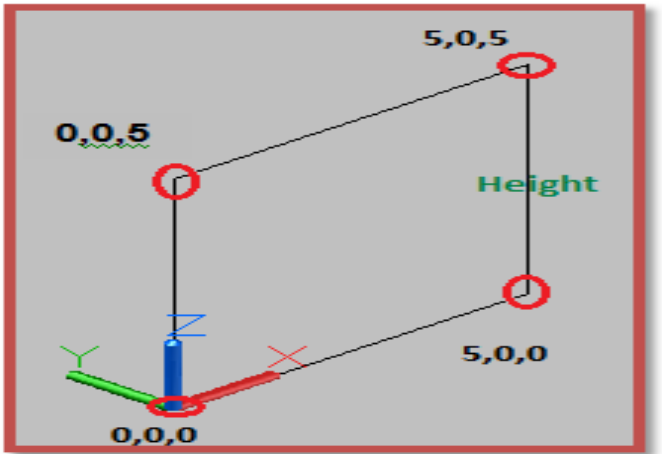
Step-2: Specify start point of polyline: 0, 0, 0

Step-3: Specify endpoint of line or [Undo]: 5,0,0

Step-4: Specify endpoint of line or [Undo]: 5,0,5

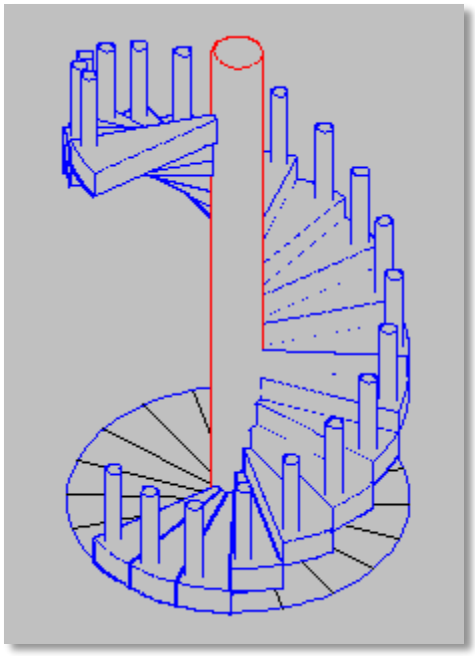
Step-5: Specify endpoint of line or [Close/Undo]: 0,0,5

Step-6: Specify endpoint of line or [Close/Undo]: 0,0,0

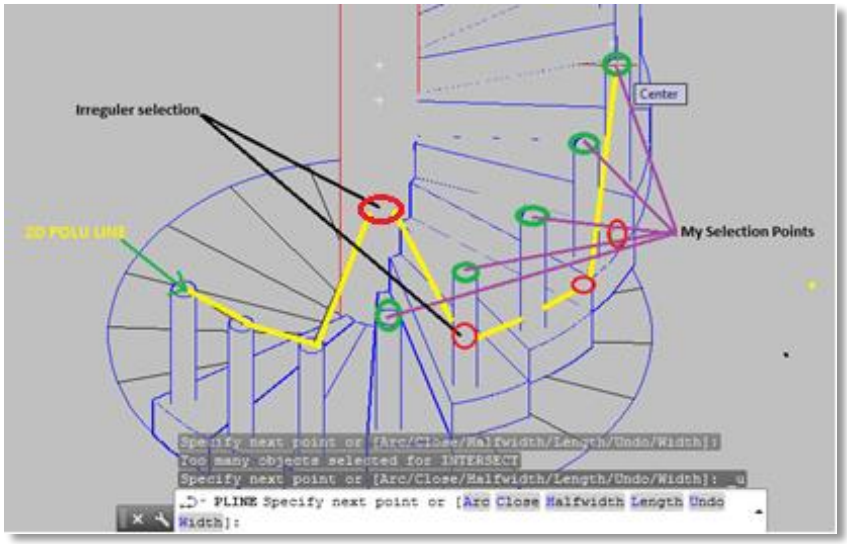


Tip: Differentiate between 2D Poly line and 3D poly line

Working with 3D objects:

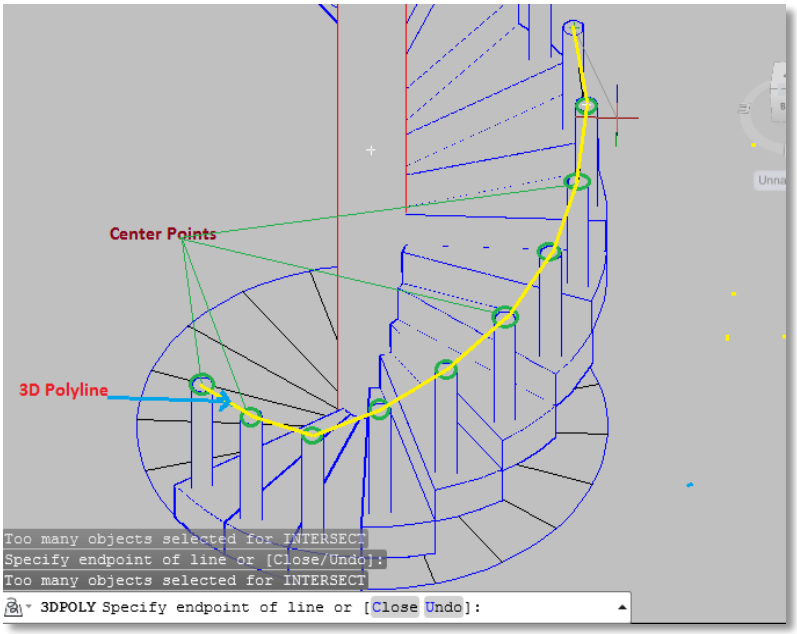


- If you try to draw a 2D poly line between the small pillars on top center selection, your selection will be irregular and not accurate as shown in the figure below



- Now If you try it with 3D poly line between the small pillars on top center selection, your selection will be regular and accurate with the center points as shown in the figure below

Note: Alignment path show a specific directions while creating an objects.



Object Tracking

Object tracking helps you to create objects at specific angles or in specific relationship to other objects with the help of alignment paths.

Object tracking also called **Auto Track** includes two tracking options:

- Polar tracking
- Object snap tracking.

1. Polar tracking

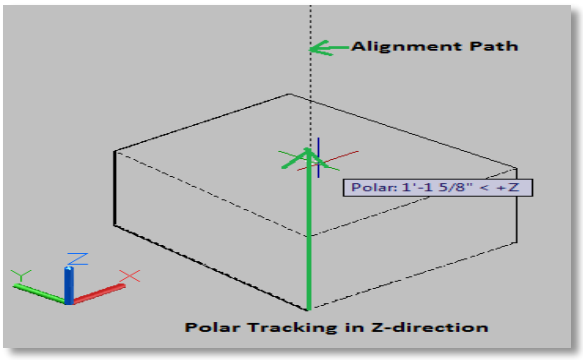
Polar tracking restricts cursor movement to specified angles. Polar Snap restricts cursor movement to specified increments along a polar angle.

When you create or modify objects, you can use polar tracking to display temporary alignment paths defined by the polar angles you specify. In 3D views, polar tracking additionally provides an alignment path in the up and down directions. In that case, the tooltip displays a +Z or -Z for the angle.

Status Bar	Button	Command Entry
Polar Tracking		F10

Procedure
Polar Tracking in Z Direction

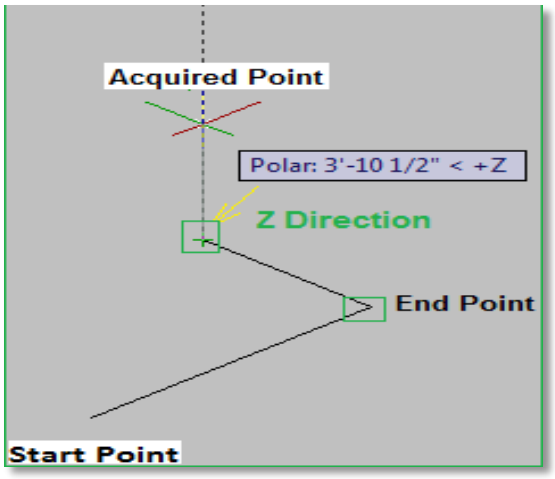
If you are working with the following 2D drawing and ready to convert it in 3D Solid by apply Press Pull option you will get the polar tracking in **Z-Direction** along with alignment path.



1. **Object snap tracking:**

   Use object snap tracking to track along alignment paths that are based on object snap points. Acquired points display a small plus sign (+). After you acquire a point, horizontal, vertical, or polar alignment paths relative to the point are displayed as you move the cursor over their drawing paths.

Status Bar	Button	Command Entry
Object Snap Tracking		F11



Move Command to Move object in Z-Direction

Move command is used to move objects by defining its displacement point

Draw Bar	Button	Command Entry
Move		m

Procedure

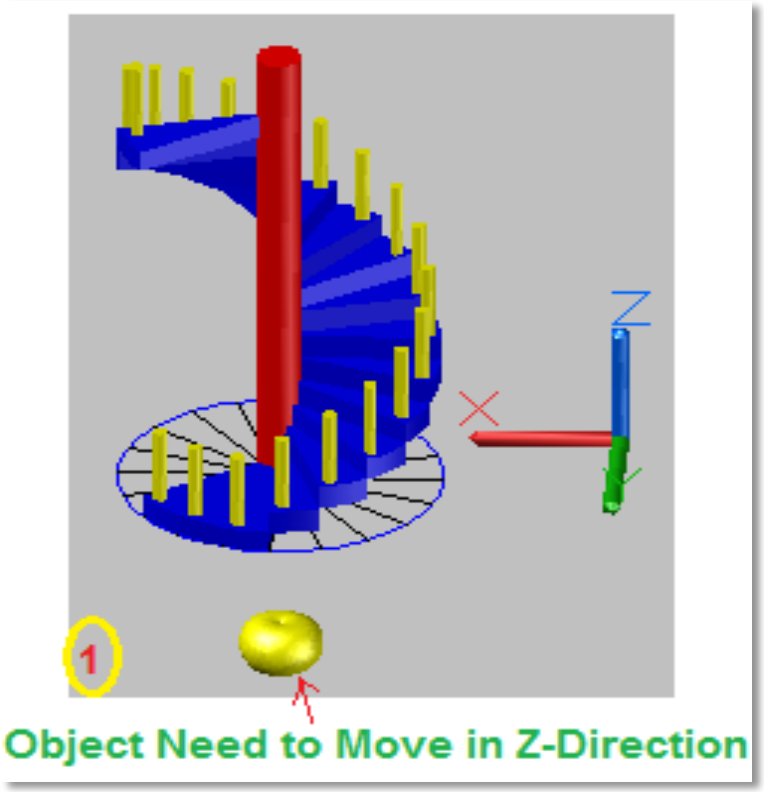
Move Object in Z Direction

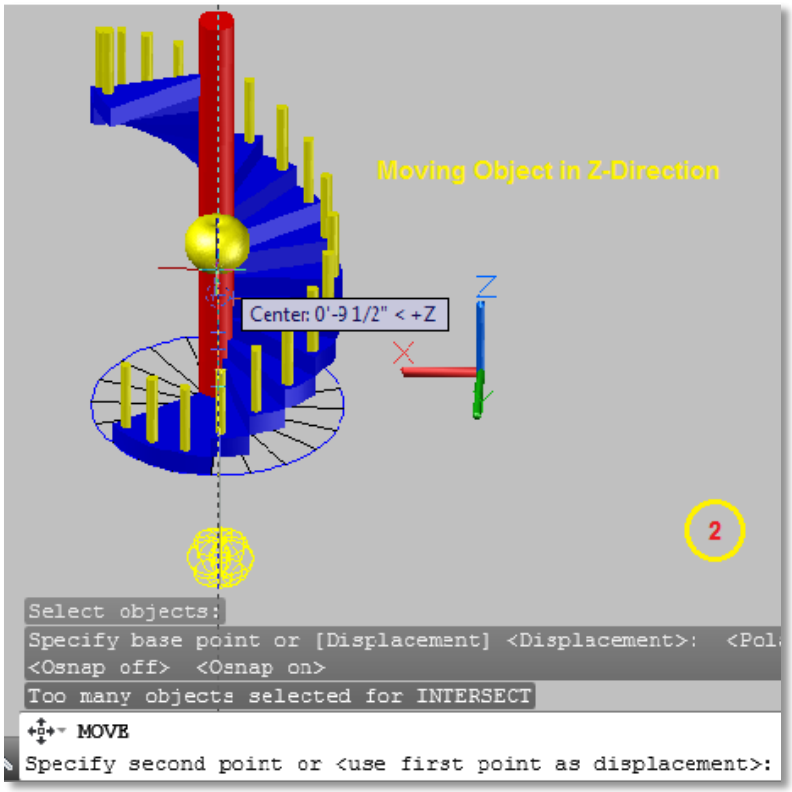
Step-1: Type **M** in command bar

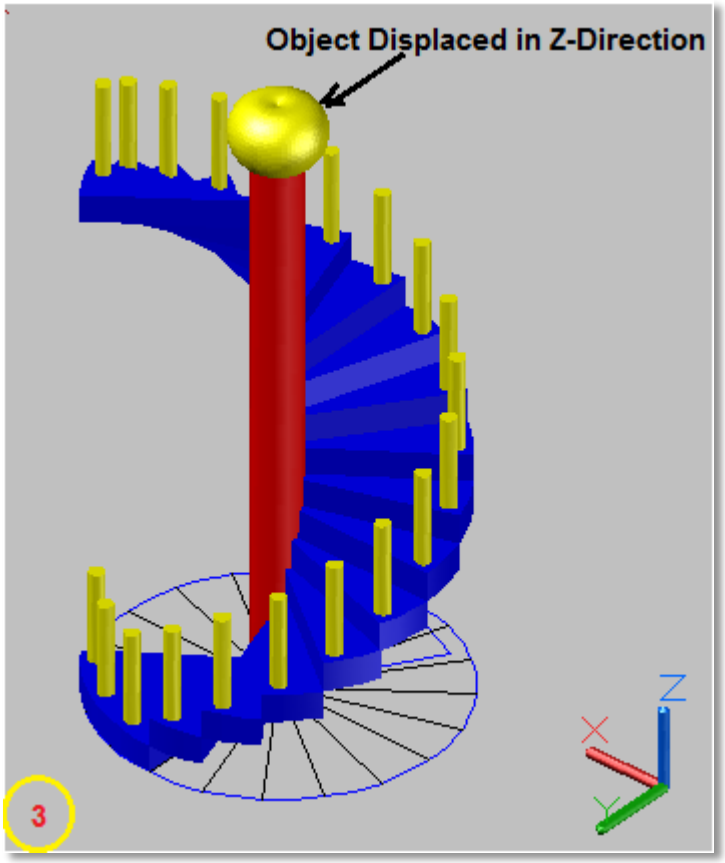
Step-2: Select objects: Select the object

Step-3: Specify base point or [Displacement] <Displacement>: Select the base point

Step-4: Specify second point or <use first point as displacement>: Select the point where to put the selected object.




Moving Object in Z-Direction
Center: 0'-9 1/2" < +Z
Z
X
Y
2
Select objects:
Specify base point or [Displacement] <Displacement>:
<Osnap off> <Osnap on>
Too many objects selected for INTERSECT
MOVE
Specify second point or <use first point as displacement>:


Object Displaced in Z-Direction
Z
X
Y
3

Acquire 3D Point Filters:

You can use coordinate filters to extract one coordinate value at a time from locations on existing objects.

Coordinate filters specify a new coordinate location by using the *X* value from one location, the *Y* value of a second location, and, for 3D coordinates, the *Z* value of a third location. When used with object snaps, coordinate filters extract coordinate values from an existing object.

1. At the prompt for a point, enter a coordinate filter (**.x**, **.y**, **.z**, **.xy**, **.xz**, or **.yz**).

   For example, enter **.x** to specify the *X* value first.

2. To extract the specified coordinate values, specify a point.

   For example, if you previously entered **.x**, the *X* value is extracted from this point.

3. At the prompt for the remaining coordinates, do one of the following:

   - Extract the remaining coordinate values by specifying a point.
   - Enter another coordinate filter.

For example, if you previously entered **.x**, specify a second point to extract the *Y* and *Z* coordinates simultaneously, or enter **.y** or **.z** to specify *Y* and *Z* values separately.

The new point location combines the coordinate values extracted from the previously specified points.

Procedure

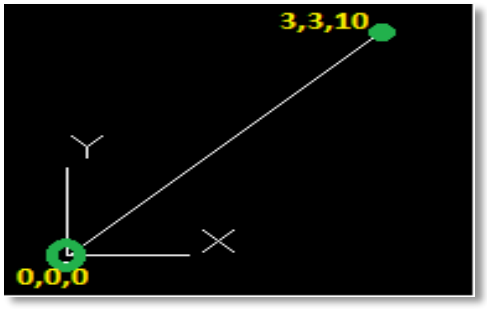
3D Point Filter

Draw a circle with XY filter to extract Z.

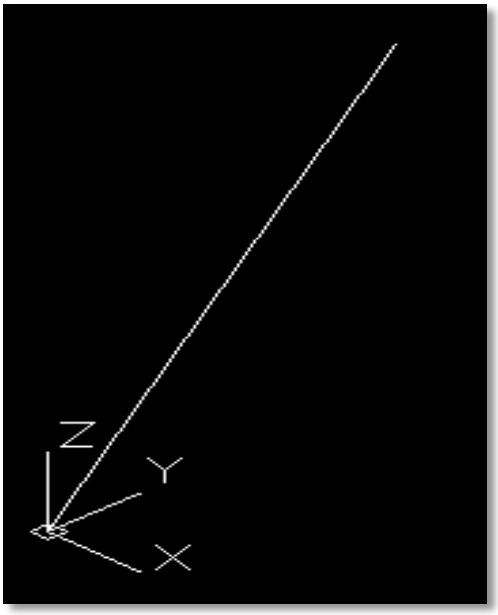
Step-1**:** Type **L** in command bar

Step-2: Specify first point: **0, 0, 0**

Step-3: Specify next point or [Undo]: **3,3,10** (with = 3, length = 3 and Height = 10)



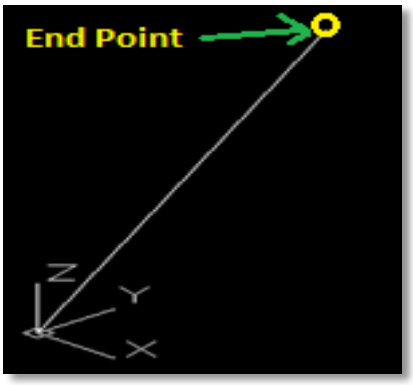
Step-4: Change the View into **SE-Isometric**



Step-5: Type a **c** for circle in command bar

Step-6: Then type Specify center point for circle or [3P/2P/Ttr (tan tan Radius)]: **.xy**

Step-7: Circle of > **3, 3** or Click the End Point of line.



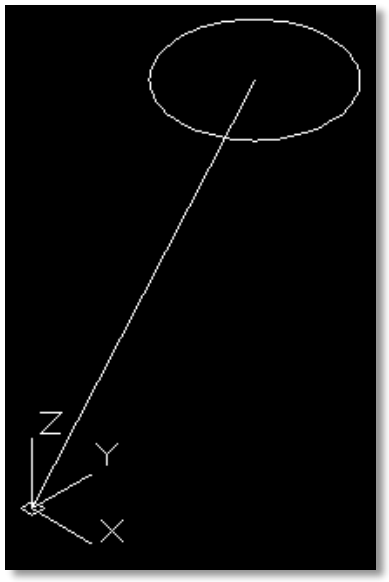
Step: 8 (need Z): **10** or Click the End Point of line.

Step: 9 Specify radius of circle or [Diameter]: **2**

Step-10: You will get the circle with center of Z point.

Learner Activity

Try 3D POINT filters to filter Z coordinate and extract XY Coordinates from it.



Helix Command

Helix mean a coil or spiral, it is an open 2D or 3D spiral.

Draw Bar	Button	Command Entry
Helix		Helix

You can use a helix as a path along which to sweep an object to create a spiral object like spring object. Threads (Internal & External)

When you create a helix, you can specify the following:

- **Base radius:** Specifies the diameter/radius of the base of the helix.
- **Top radius** : Specifies the diameter/radius of the top of the helix
- **Height :** Specifies the height of the helix
- **Number of turns:** Specifies the number of turns (revolutions) for the helix. The number of turns for a helix cannot exceed 500.
- **Turn height**: Specifies the height of one complete turn within the helix.
- **Twist direction:** Specifies whether the helix is drawn in the clockwise (CW) or the counterclockwise (CCW) direction. The default value for the helix twist is CCW.

Procedure

Remember

If Base Radius and top Radius are same, then it is called cylindrical helix.

Apply Helix to make a spring with Sweep Command

Step-1: Type Helix command in command bar.

Step-2: Specify center point of base: **Define any center point**

Step-3: Specify base radius or [Diameter]: **3**

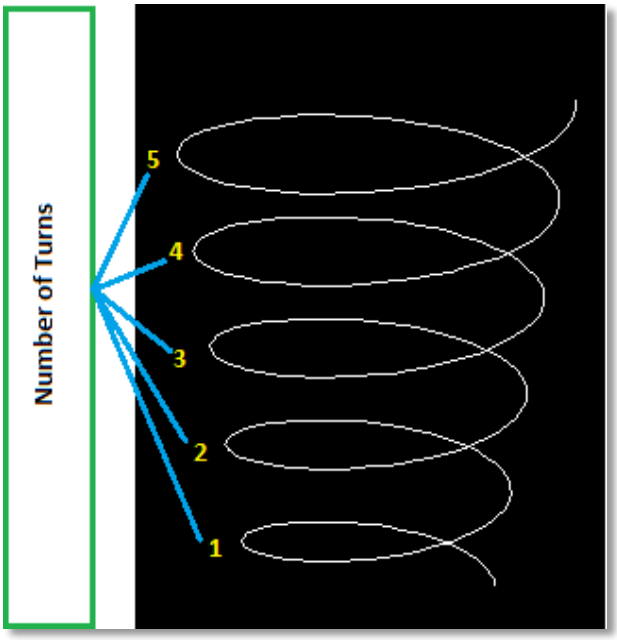
Step-4: Specify top radius or [Diameter] <0'-3">: **5**

Step-5: Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <1'-0">: **t**

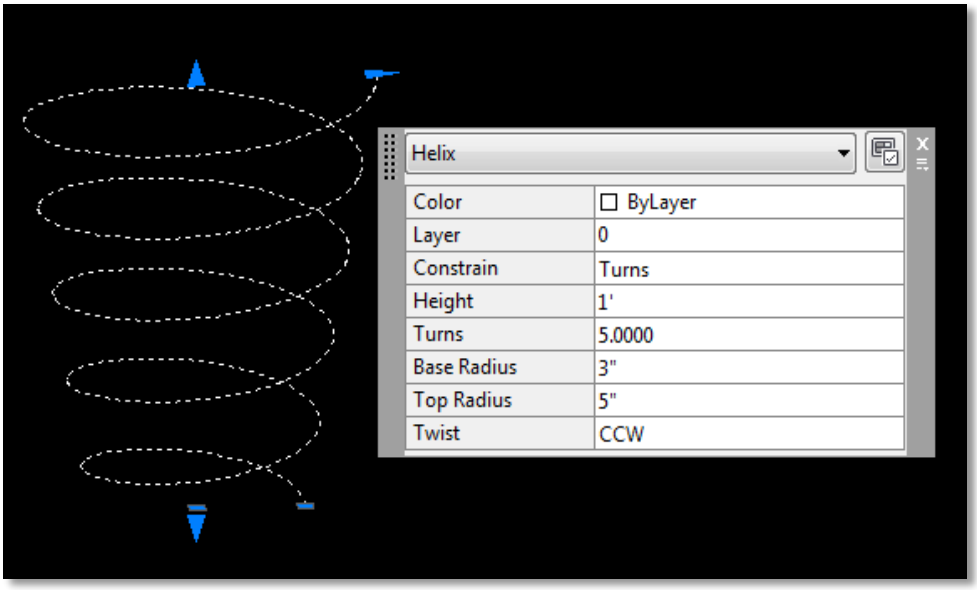
Step-6: Enter number of turns <3.0000>: **5**

Step-7: Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <>: **12**

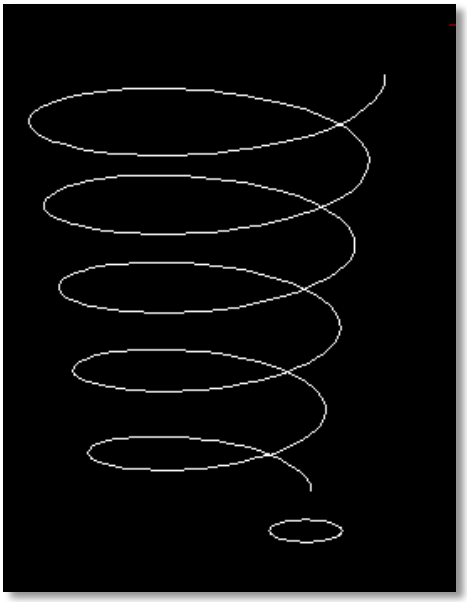
Step:-8 you should see the object as shown in the figure below.



Note: You can modify Number Turns, height, twist etc. by using helix properties



Step-9: Draw a circle with radius of 0.6 inches at the base of helix as shown in the figure below.

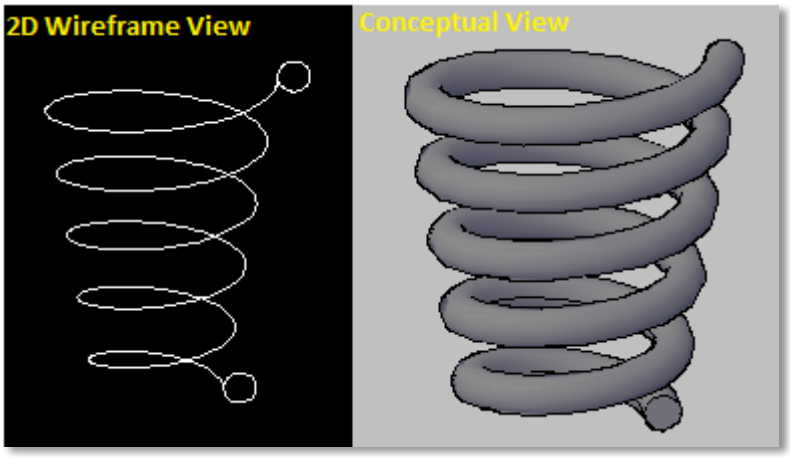


Step-10: Apply Sweep command by typing **sweep** in command bar

Step-11: Select objects to sweep or [MOde]: **<Select the circle>**

Step-12: Select sweep path or [Alignment/Base point/Scale/Twist]: **<Select the Helix>**

Step-13: You will see the final object as shown in figure below.



Learning Unit 2: Define User Coordinates System

Overview

In This unit you will learn the user coordinates system in detail, after completing this module you will be able to configure User Coordinates System (UCS) properties, differentiate different icons of different UCS and apply different functions to UCS.

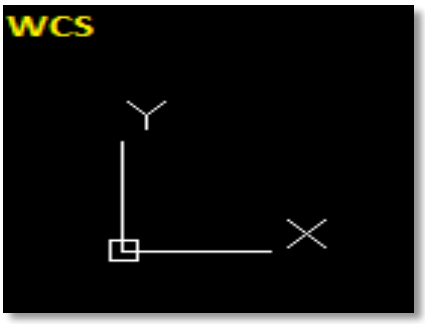
Further you will be able apply restore a saved and named UCS with **"R"** key and executes the UCS dialog box using UCSMAN and generating different views through View Cube.

1. Orientation of the User Coordinate System (UCS)

User Coordinate System provides a visual cue to how we're looking and viewing at the drawing, and it changes when we're looking from a 3D perspective.

The UCS icon, an essential part of drawing, is found in the lower-left corner of the drawing area and specifies the positive coordinates for the x-axes, y-axes, and z-axes. AutoCAD displays the UCS icon differently for 2D, 3D environments.

Take a look at the icon as you can find it initially located at the origin when startup AutoCAD application. The two lines in the icon show the positive X and Y directions of the User Coordinate System (UCS) called World Coordinate System (WCS), which is the default system for all AutoCAD drawings. You'll change the orientation of the icon to the drawing and then change the orientation of the drawing to the screen.



Do you know!

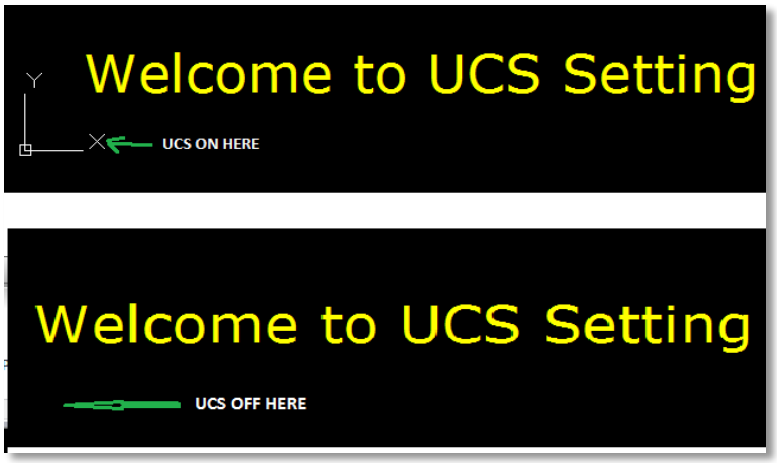
You can turn ON/OFF UCS icon by using **UCSICON** command.

Procedure

ON/OFF UCS Icon

Step-1: Type **UCSICON** in the command bar Press **Enter**

Step-2: Enter an option [ON/OFF/All/Noorigin/ORigin/Selectable/Properties] <ON>: **OFF**

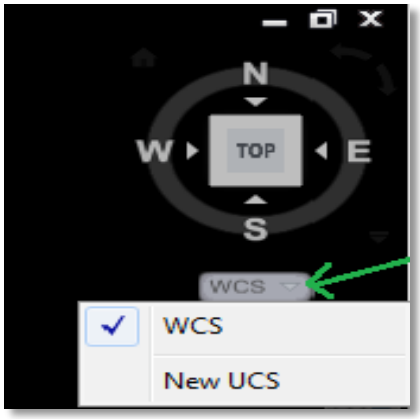


Sr.	UCS ICON Option	Description
1	ON:	Displays the UCS icon.
2	OFF:	Turns off the display of the UCSICON.
3	All Affects:	The display of the UCSICON in all viewports.
4	Noorigin:	Always displays the UCS at the lower left corner.
5	Origin :	Shows the UCS at the 0,0,0 origin of the current UCS.
6	Properties	Changes the display properties of the UCS icons(s).

2. Multiple Switches of “UCS”

You can set different USC by using **ucs** command, or can be set by Using the **ViewCube** found in the upper-right corner of the drawing area, click the UCS drop-down menu and select New UCS, as shown in Figure below.

The UCS defines the positive X and Y directions in your drawing. A drawing can have several UCSs, but can use only one at a time as your current UCS as per the drawing to work with.



3. Multiple Switches Comprise in "UCS"

Multiple switches are:

- **Face:** Aligns the UCS to the selected face of a solid object.
- **Named:** Saves or restores a UCS.
- **Object:** Lets you define a new UCS by pointing at an object.
- **Previous:** Restores the previous UCS.
- **View:** Establishes a new UCS whose XY plane is perpendicular to your viewing direction (e.g. parallel to your screen).
- **World :** Restores the world UCS
- **X/Y/Z:** Rotates the UCS around a specified axis.
- **New :** Defines a new coordinate system by one of six methods: **Origin, Z Axis, 3 Point, Object, Face, View X, Y, Z**

Procedure

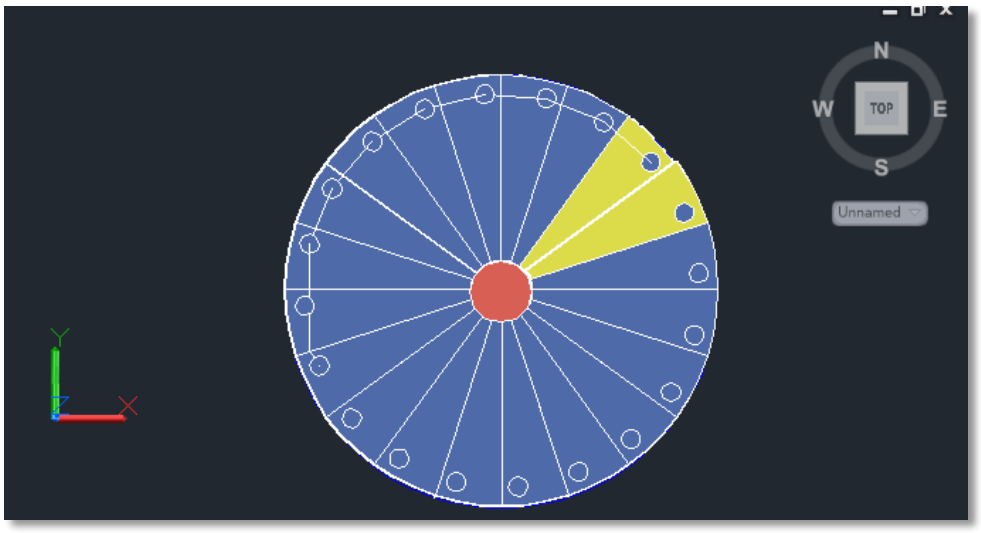
How to Set UCS

Step-1: **Can be set**

- By Using **ViewCube** just click the arrow.
- By using Command just type **ucs** in command bar.

Current view of the drawing is shown in figure below

Current ucs name: *TOP*



Step-2: Specify origin of UCS or [Face/NAmed/OBject/Previous/View/World/X/Y/Z/ZAxis] <World>: **z** <Press Enter>

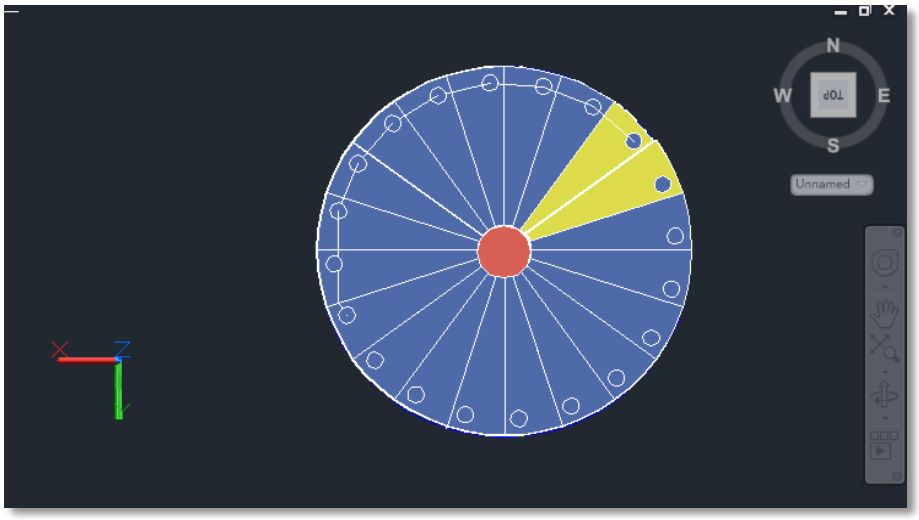
PLAN

Plan command is used to select Current UCS or to switch between other UCS and world coordinates.

Step-3: Specify rotation angle about Z axis <90>: 180 <Press Enter>

The VIEWCUBE shows the TOP view mirror to the original UCS and coordinates show X and Y in opposite direction as shown in the figure, if you want to reorient the drawing according to your last created UCS then following from step-4

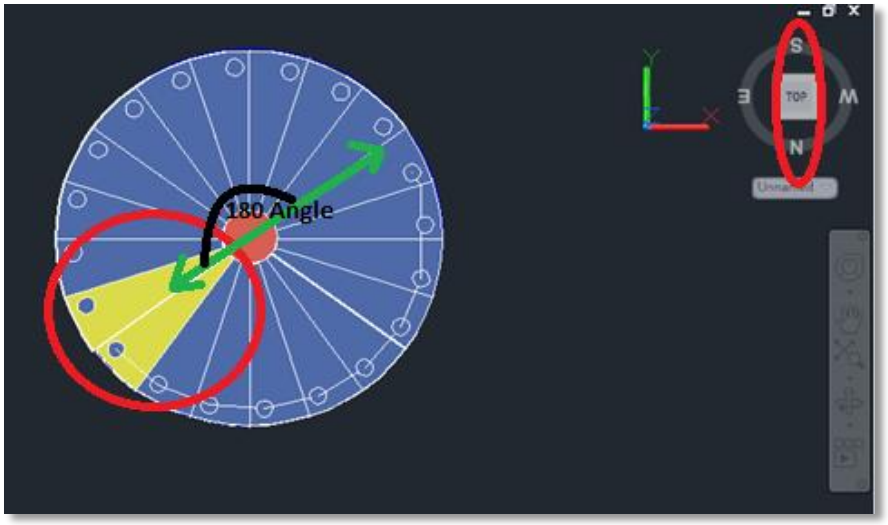
You can used the UCS command to reorient the UCS icon relative to the drawing.



Step-4: Type **PLAN**

Step-5: Enter an option [Current ucs/Ucs/World] <Current>: **c**

You should get the drawing according to your created UCS in current UCS, ViewCube changes the positions of North and south as shown in figure below.

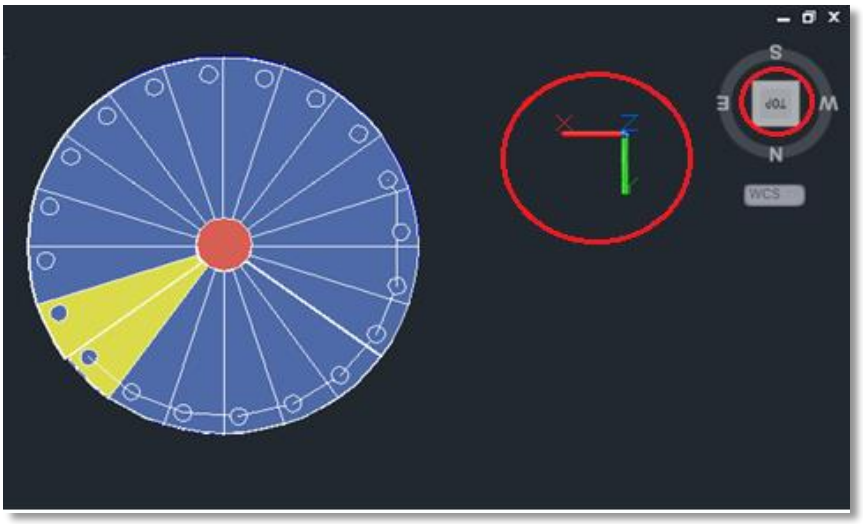


Note: The positive X and Y directions of the current UCS are directed to the right and upward, respectively. This process is a little bit like turning your monitor upside down to get the correct orientation.

Step-6: **To restore WCS** : Type ucs press <Enter>

Step-7: Specify origin of UCS or [Face/NAmed/OBject/Previous/View/World/X/Y/Z/ZAxis]

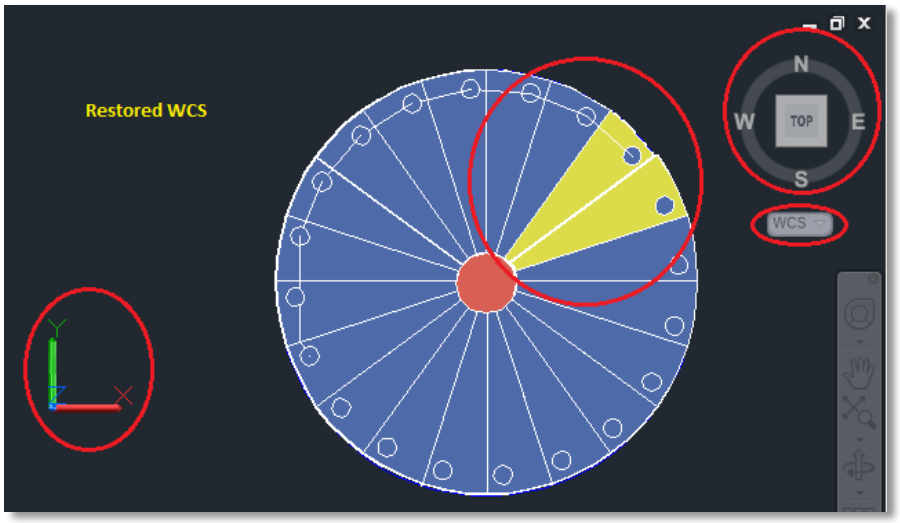
<World>: **w**



Step-8: Type **PLAN** Press<Enter>

Step-9: Enter an option [Current ucs/Ucs/World] <Current>: **w**

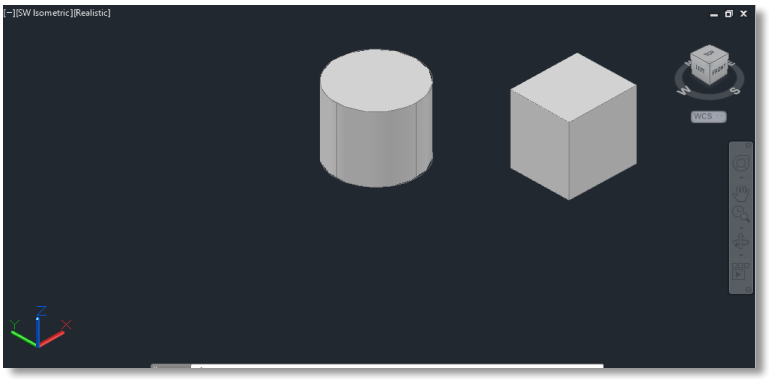
You will restored the WCS as shown in the figure below



Procedure

How to Use UCS Option

This Procedure described the complete procedure how to name UCS, restore UCS and delete UCS by applying Face and Object UCS to the following object shown in the figure below.

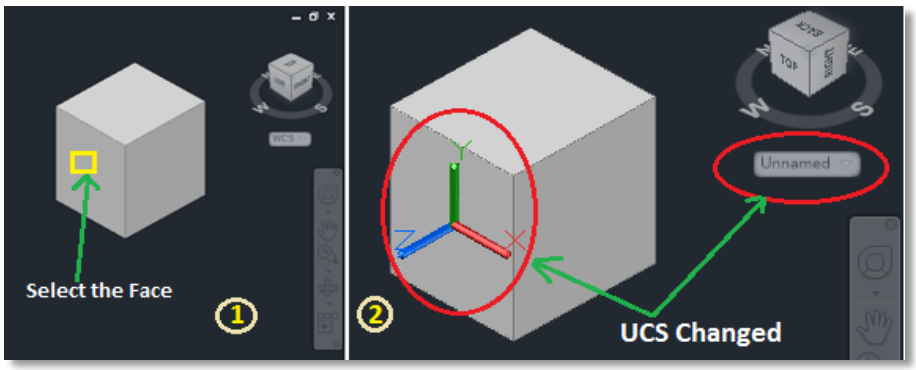


Step-1: type **UCS** in command bar

Step-2: Specify origin of UCS or [Face/NAmed/OBject/Previous/View/World/X/Y/Z/ZAxis] <World>: **F**

Step-3: Select face of solid, surface, or mesh: **Select Face** as shown in figure below

Step-4 Enter an option [Next/Xflip/Yflip] <accept>:**<Enter>**



Step-5: Now Name the UCS→ Type **UCS** command in command bar

Step-6: Specify origin of UCS or [Face/NAmed/OBject/Previous/View/World/X/Y/Z/ZAxis] <World>: **na**

Step-7: Enter an option [Restore/Save/Delete/?]: **s**

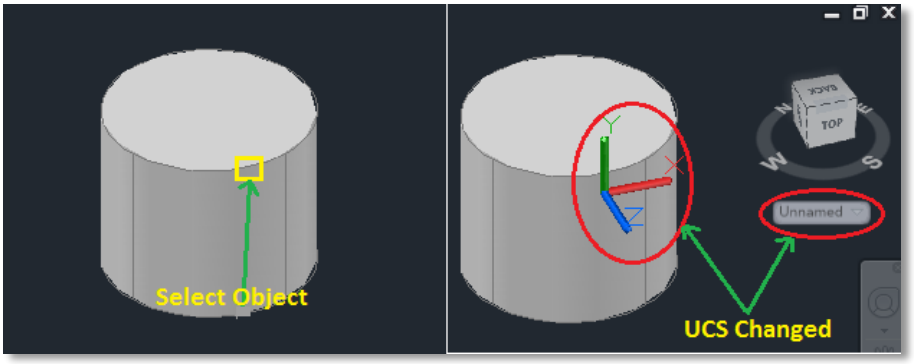
Step-8: Enter name to save current UCS or [?]: **face-ucs**

Now use UCS for an Object Option and save it with other name

Step-1: type **UCS** in command bar <Enter>

Step-2: Specify origin of UCS or [Face/NAmed/OBject/Previous/View/World/X/Y/Z/ZAxis] <World>: **ob**

Step-3: Select object to align UCS: **Select object** as shown in figure below



Step-5: Now Name the UCS→ typr **UCS** command in command bar

Step-6: Specify origin of UCS or [Face/NAmed/OBject/Previous/View/World/X/Y/Z/ZAxis] <World>: **na**

Step-7: Enter an option [Restore/Save/Delete/?]: **s**

Step-8: Enter name to save current UCS or [?]: **object-ucs**

Now use UCS to restore the UCS name face-ucs

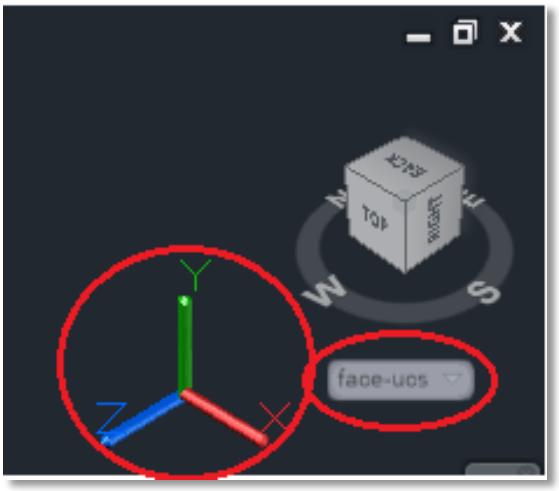
*Step-1: Type **ucs** command in command bar*

*Step-2: Specify origin of UCS or [Face/NAmed/OBject/Previous/View/World/X/Y/Z/ZAxis] <World>: **na***

Step-3: Enter an option [Restore/Save/Delete/?]: ***r***

Step-4: Enter name of UCS to restore or [?]: ***face-ucs***

You will get the following UCS



Learner's Activity

- Follow the above procedure to restore the UCS named **object-ucs** previously created and restore **WCS** as well.
- How to list the UCS if you have created one or more UCS in your drawing?
- How to delete UCS?

4. Dynamic UCS

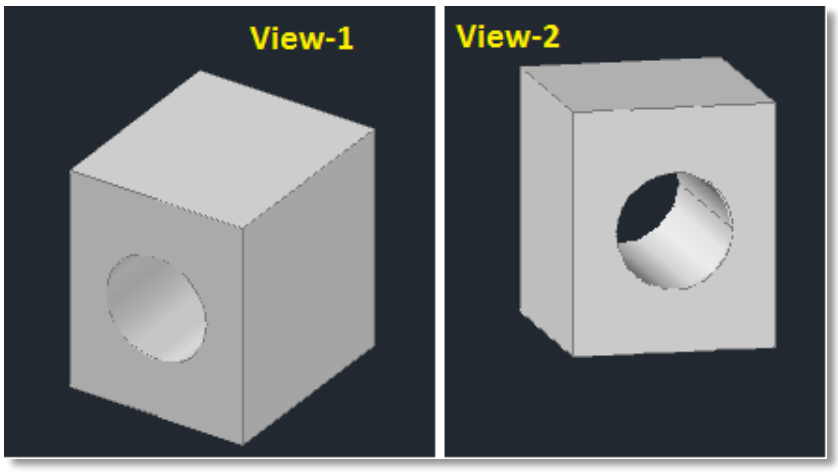
Let's you temporarily and automatically align the XY plane of the UCS with a plane on a solid model while creating objects.

You can use the dynamic UCS to create objects on a planar face of a 3D solid without manually changing the UCS orientation.

Procedure

Use of Dynamic UCS

Create the following object



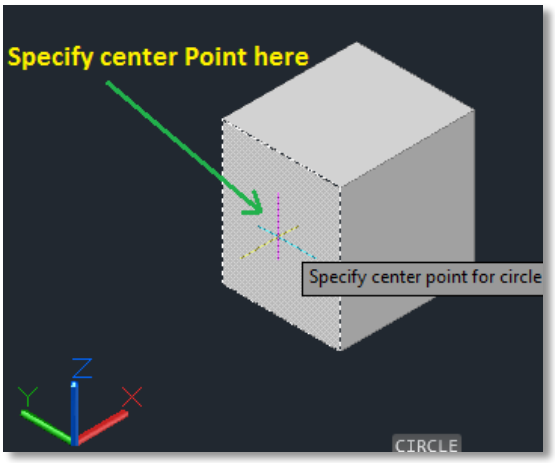
Step-1: Draw a box

Step-2 **ON** the **Dynamic UCS** by the following three method

1. **Ctrl+D**
2. **F6**
3. **Click** **to On/off**

Step-3: Type **c** for circle command

Step-4: Specify the center point for circle where the UCS indicated as shown in the figure below.

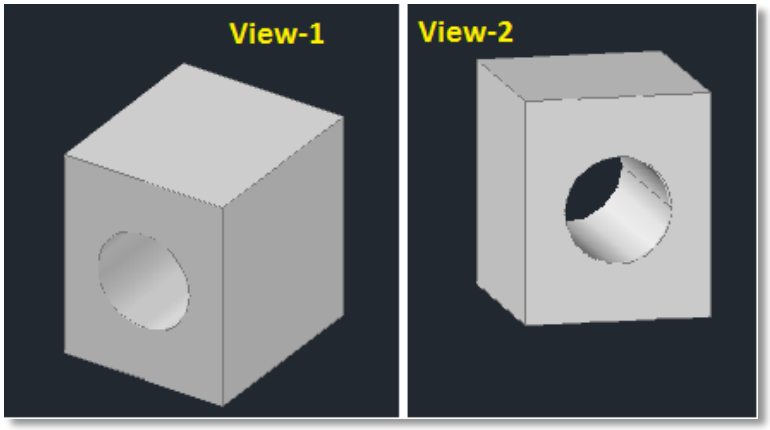


Step-5: Specify radius of circle or [Diameter] <0'-9 1/8">: **10-inches**

Step-6: Extrude the object

Step-7: Subtract the last extruded object from the box

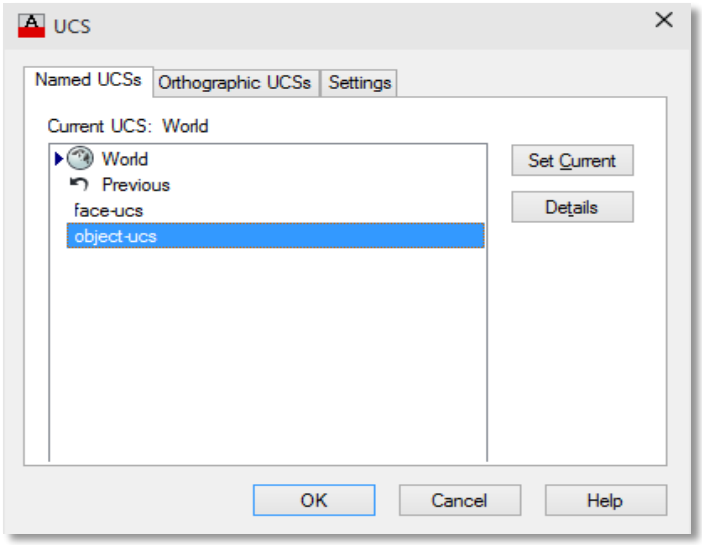
You will get the following



Use of "UCSMAN" Command:

UCSMAN Command is used to open the UCS dialog box. Alternatively, you can click the UCS, Named UCS button on the Coordinates panel of the View tab.

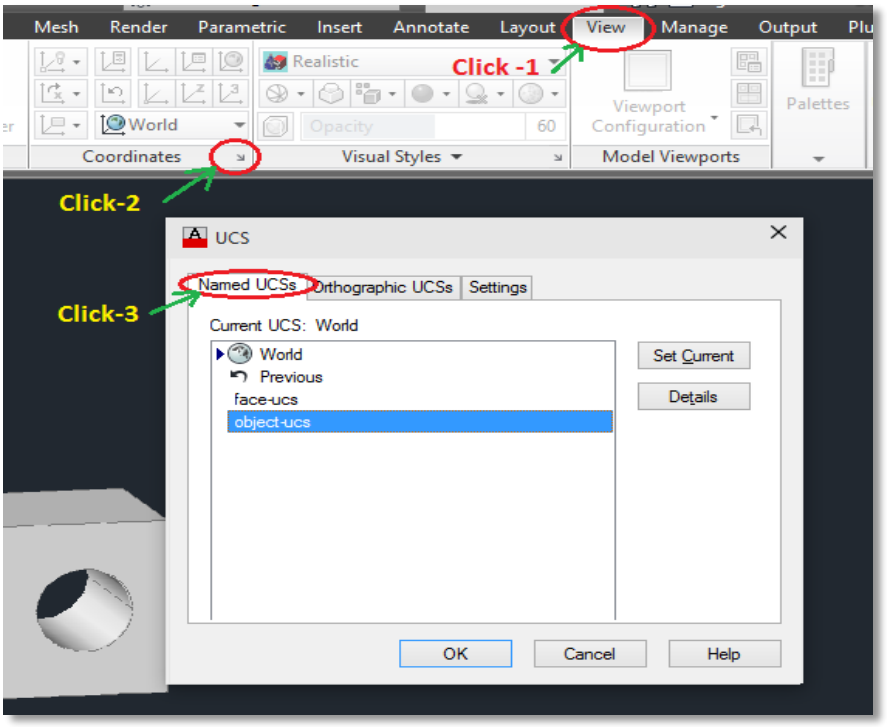
Method-1: Enter UCSMAN in command bar, you will get the following dialog box



From the UCS dialog box, click the Named UCSs tab, and then click the UCS name (**Object-ucs**) to set as a current UCS.

You can also apply more properties from this dialog box, just Right Click on any of the UCS→ you will get rename, delete and set current UCS.

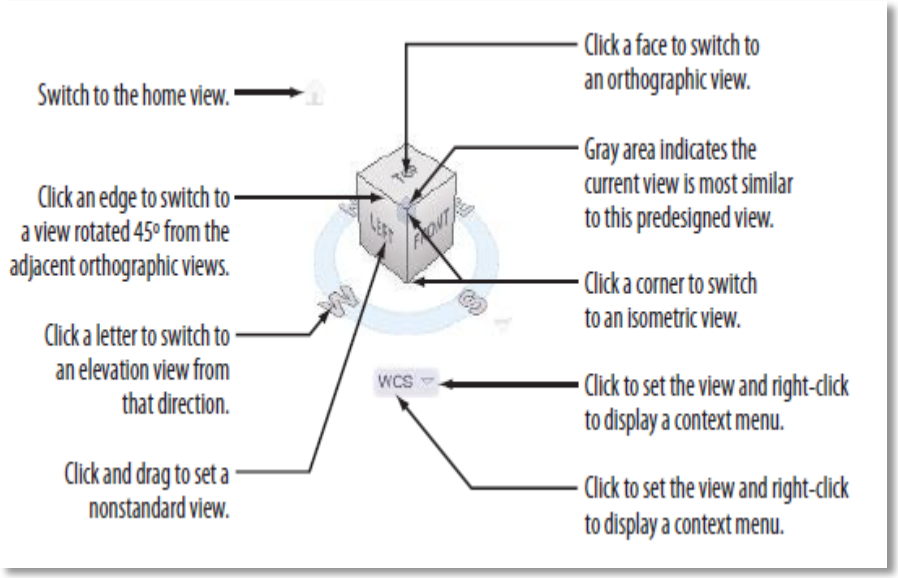
Method-2: You can click the UCS, Named UCS button on the Coordinates panel of the View tab as shown in the figure below.



Viewcube:

To change the view orientation by using the ViewCube. Located in the upper-right corner of the drawing area, is used to access common views quickly, return to a saved view, or navigate freely in the drawing area and to define New UCS or to switch between different UCSs.

You've already used the ViewCube to get different visual feedback in this module and in previous module as well.



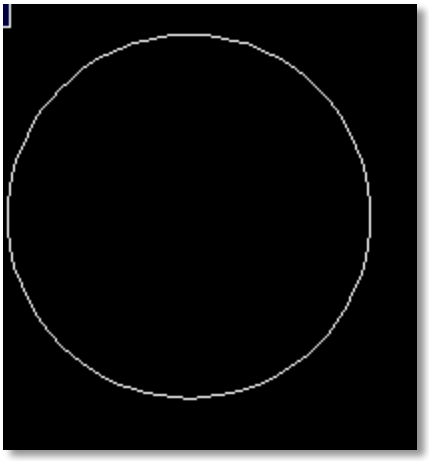
Project

Making a Screw NUT

STEP BY STEP Operations on AutoCAD

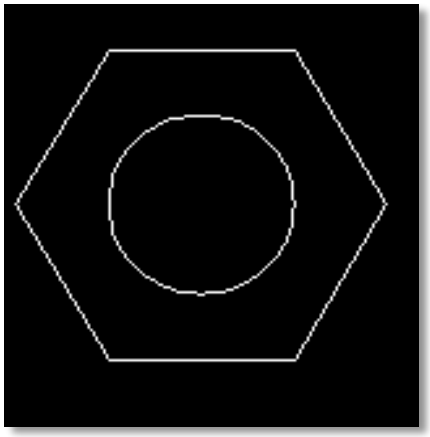
GET READY. Before you begin these steps, be sure to set the units.

Step-1: Draw a circle with radius 3 units.



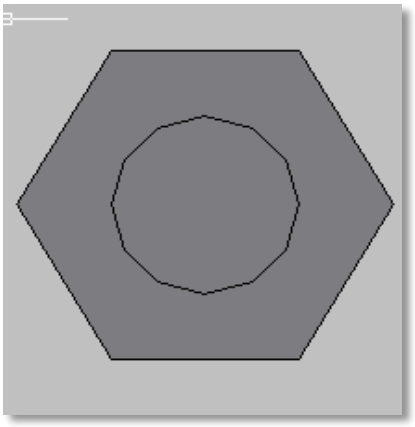
Step-2: Draw a polygon of six sides with same center point with radius of 6 units

Note:
Polygon must be inscribe in circle

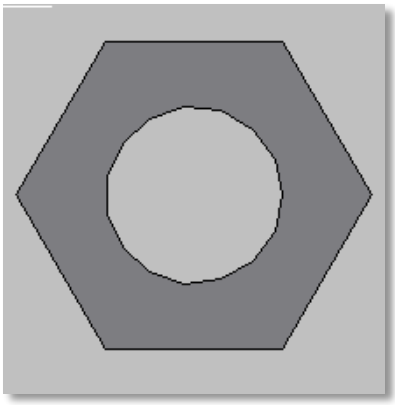


.

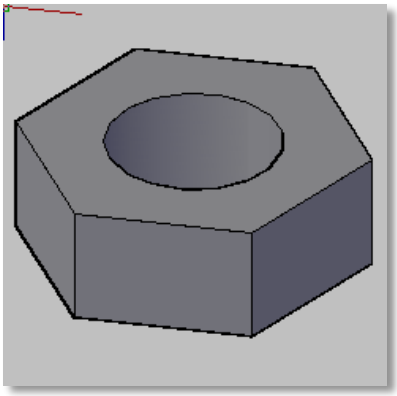
Step-3: Use region command, to create one object.



Step-4: Give subtract command to subtract the circle form the polygon.

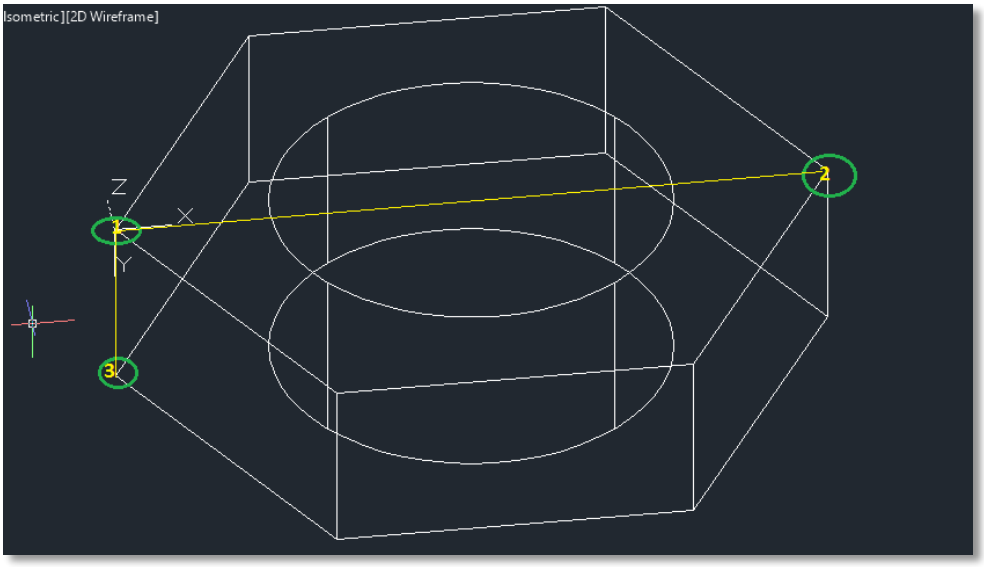


Step-5: Now extrude the object to 4 units.



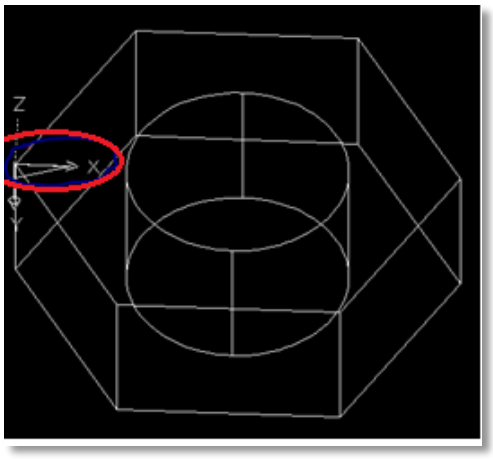
Step-6:

- Change the view to 2D Wireframe
- Type UCS and set the new UCS as shown in the figure below



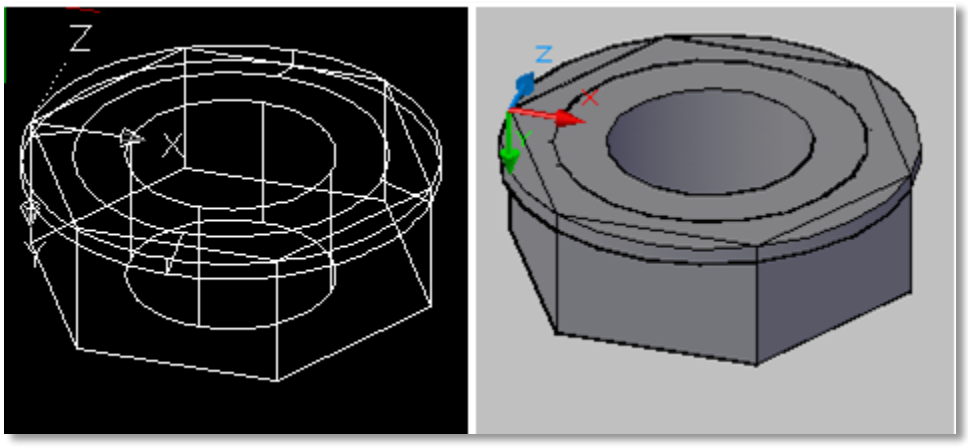
Step-7:

- Enter **Pl** for polyline
- Select the UCS origin with **ortho on** function and make a line of **1.5** units in **X-axis** direction.
- Then turn off the ortho, give sub command of (tt) to mark a temporary Tracking point on back to the origin.
- Now from the origin draw line with extension of **0.50** units in **Y-axis** direction
- You will get a triangle as shown in figure below.



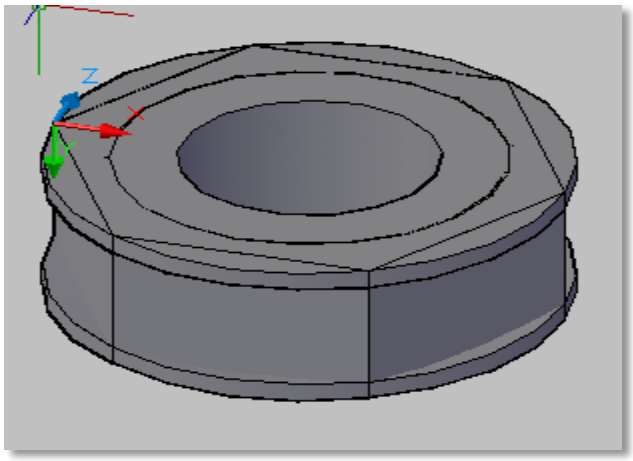
Step-8:

- Use revolve command
- Select the triangle
- Define axis as a center of lower circle to the center of upper circle with angle of 360.

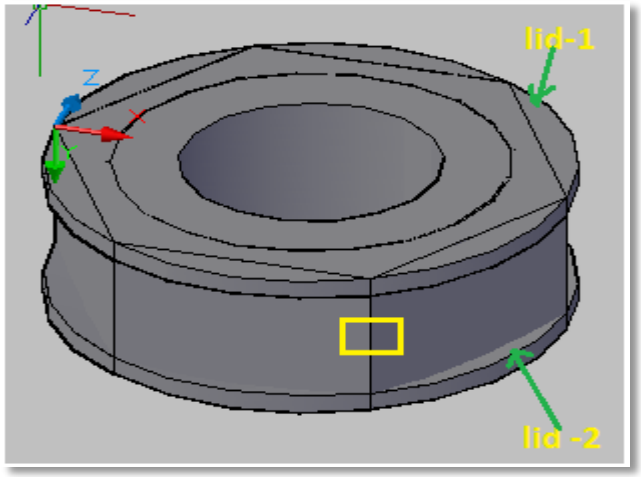


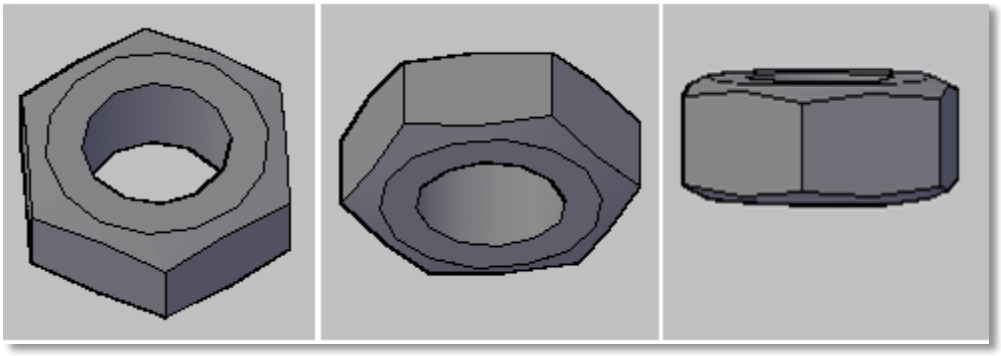
Step-9:

- Use mirror command
- Make a copy of the last revolved object at the lower side as shown in the figure below.



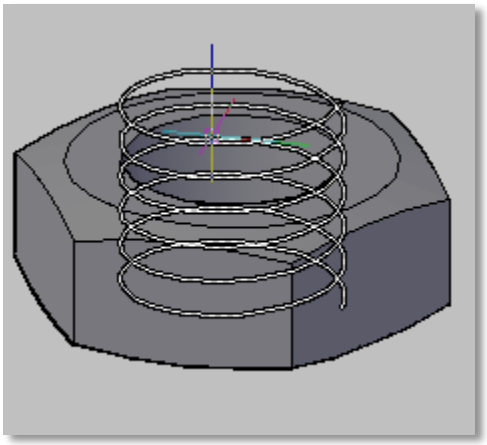
Step-8: Use subtract command to subtract Lid-1 and lid-2 from main object



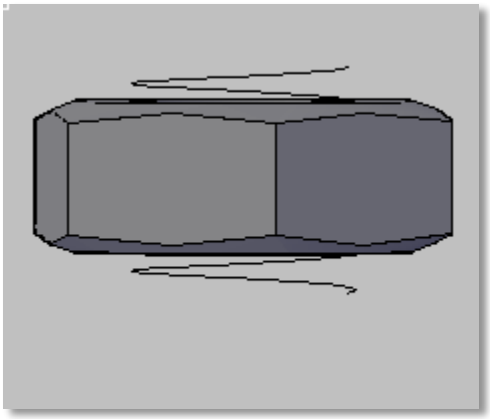


Step-9:

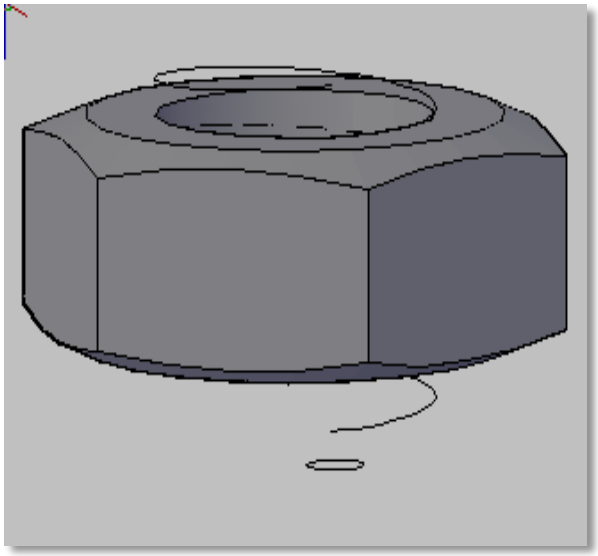
- Use helix command
- select the center of the nut
- Base and top radius should be 3 units equal to the radius of the nut
- Enter sub command of turns (**t**)
- Enter Number of Turns 6
- Enter height of Helix 6



Step-10: Move the helix downwards to look like as shown in the figure below



Step-11: Draw a circle with radius of 0.6 units at the bottom as shown in the figure

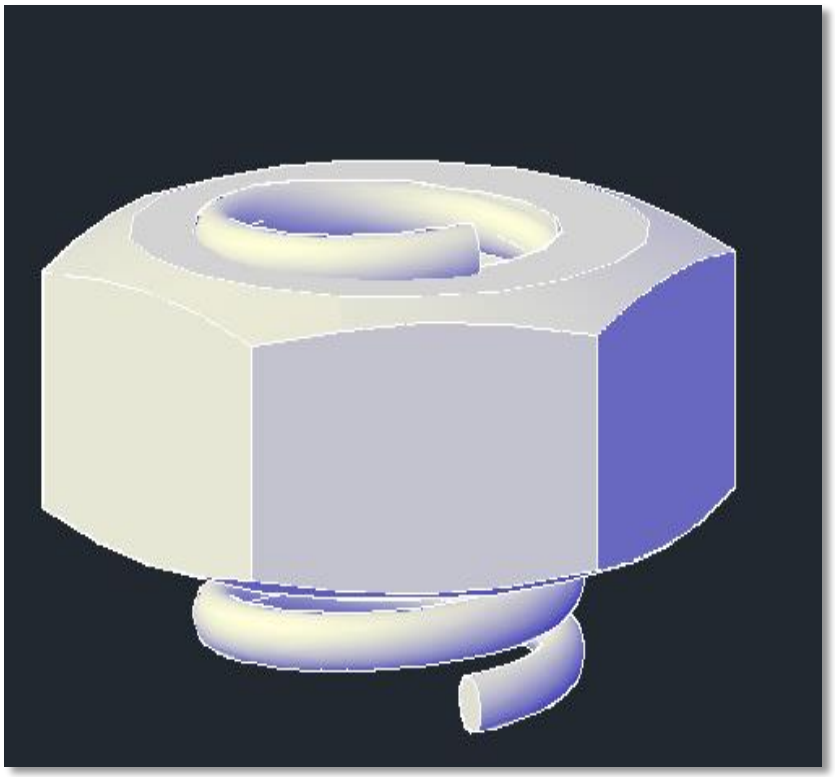


Note:

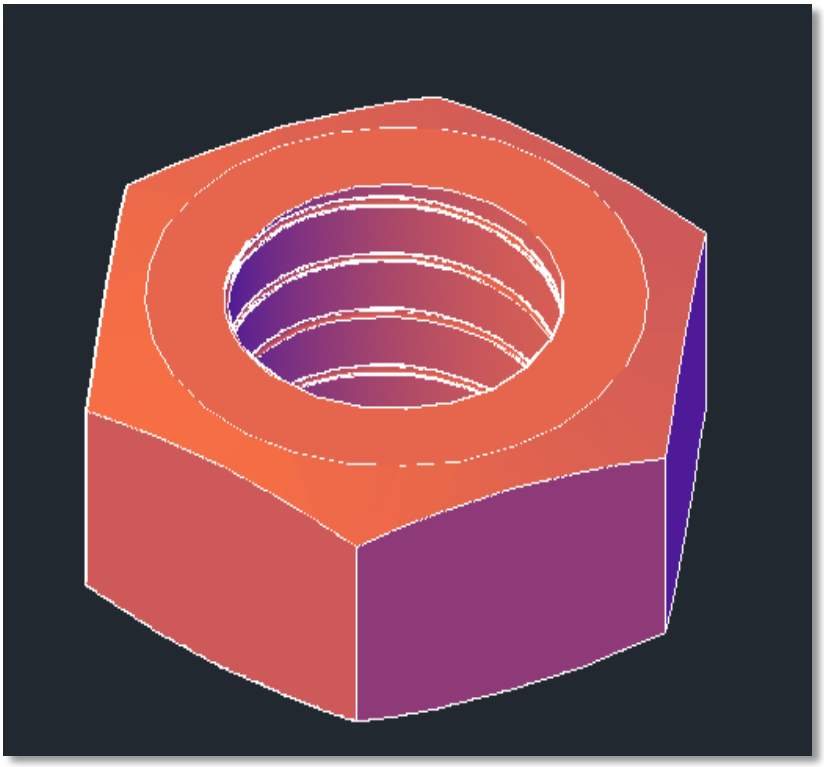
- Draw triangle for V-Thread
- Draw square for square Thread

Step-12:

- Enter sweep command
- Select the circle enter
- Then select the helix



Step-13: Now subtract the spring from the NUT and get the following object as shown in figure below.



Summary of Module

- In this module we learnt the basic of 3D User and Z Cartesian (X, Y, Z) coordinates system with the help of 3DPOLY command and object Tracking tools including Polar tracking and object snap tracking to work in a specific path and specific angle while developing an objects . We also used acquiring 3D point filter to extract one coordinate value from location on existing objects.

- The core objective we covered in this module is the User Coordinate System (UCS) for the orientation of the objects in the 2D and 3D drawing. We learnt from the two lines positive X and positive Y axis of the user coordinate that is called the World Coordinate System to Three line X- , Y- and Z-Axis Cartesian coordinate system including multiple switches FACE, Named, Object, Previous, View, World, X/Y/Z and New to set different UCS for an object in the drawing to plan our current, world or user define coordinates system by using **PLAN** command.

- We also learnt the Viewcube tools along with the User Coordinate System located in the Upper-Right Corner of the drawing area to navigate freely in the drawing area and to change the view orientation between different UCSs.

Frequently Asked Questions (FAQs)

FAQ 1:	Can we use different UCS at one time?
Answer	NO! A drawing can have several UCSs, but can use only one at a time.
FAQ 2:	Two lines X Positive and Y Negative belongs to which Quadrant?
Answer	Quadrant-IV
FAQ 3:	How can you get 3DPOLYINE other than discussed in unit-1 of this module?
Answer	Switch to Workspace3D Modeling→Draw→Polyline Icon
FAQ 4:	Is there any Pick Point in Line?
Answer	Yes! There are Three Points to Pick a line i.e. Start , Mid and End Point
FAQ 5:	If you enter First point X=0, y= 0 and z = 5, and second point x = 5, y =10 for polyline, then what will be the value of Z at the second Point?
Answer	The value of the Z will be remain 5
FAQ 6:	If you are working with WCS as a current view, is there any need to save the WCS?
Answer	No! The WCS is a permanent part of all drawings, so you never need to save it.
FAQ 7:	How can we set the increment angle of Polar tracking?
Answer	Right Click on Polar Tracking Icon and the set Polar angle.
FAQ 8:	What is the default increment angle of Polar tracking?
Answer	The default increment angle of Polar Tracking is 90.
FAQ 9:	How can you Turn ON/OFF Midpoint option for a line?
Answer	Right Click on OSNAP Icon and the turn ON/OFF Midpoint option.
FAQ 10:	How can you delete a user define UCS?
Answer	1. Type UCS command in command bar 2. Specify origin of UCS or [Face/NAmed/OBject/Previous/View/World/X/Y/Z/ZAxis] <World>: na 3. Enter an option [Restore/Save/Delete/?]: D

Test Yourself!

Please mark the correct one from the given options. You can check your answer with the Answer Key at the end of this module.

1. Short cut key for Polar Tracking is?

a. F3 b. Ctrl + d

c. F10 d. F11

2. Short Cut for Turn ON/OF Dynamic UCS?

a. Ctrl + f b. Ctrl + D

c. F8 d. None

3. *Cartesian coordinate system define in*

a. 1900s b. 1600s

c. 1800s d. None

4. In 2D Cartesian coordinates system, you specify points on the *XY* plane, also called the?

a Work Plane b 2D Plan

c Block Plan d None

5. Which command is use to open and set UCS dialog box?

a UCSMAN b UCSICON

c UCS d Non

Answer Key

MCQ Number	Correct Answer
1	c
2	b
3	b
4	a
5	a

AUTOCAD

Module-5

LEARNING GUIDE

National Vocational Certificate Level 3

Version 1 - July 2013

Module 5: Draw 3D Orbit, Navigation and Model

Learning Outcomes

After completion of this learning module, you will be able to:

- Develop Familiarity with 3D Orbit
- Perform 3D Navigation
- Operate 3D Object

Learning Unit 1: Develop Familiarity with 3D Orbit

Overview

This unit describes familiarity with AutoCAD 3D Orbit and its different types. After completion of this learning unit, you will be able to work with "**3DOrbit**" command for constrained orbit on selected object and on whole drawing as well. You will also be able to provide due assistance in developing zoom (Extends, window etc.) and pan facility and apply different projection mode Parallel and Perspective option with respect to fly and walk orbiting to explore the use of orbiting.

Further you will be able to select different visual styles (e.g. 3D Hidden, 3D Wireframe, Conceptual, and Realistic), different visual aids (e.g. Compass, Grid and UCS Icon) and Preset Views for different orbiting like 3D Orbit, Free orbit and Continuous orbit.

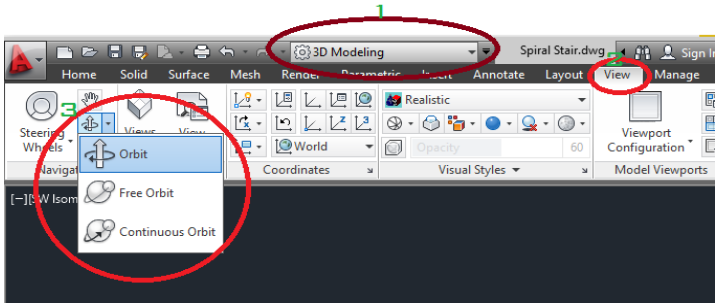
1. Introduction to 3D Orbit

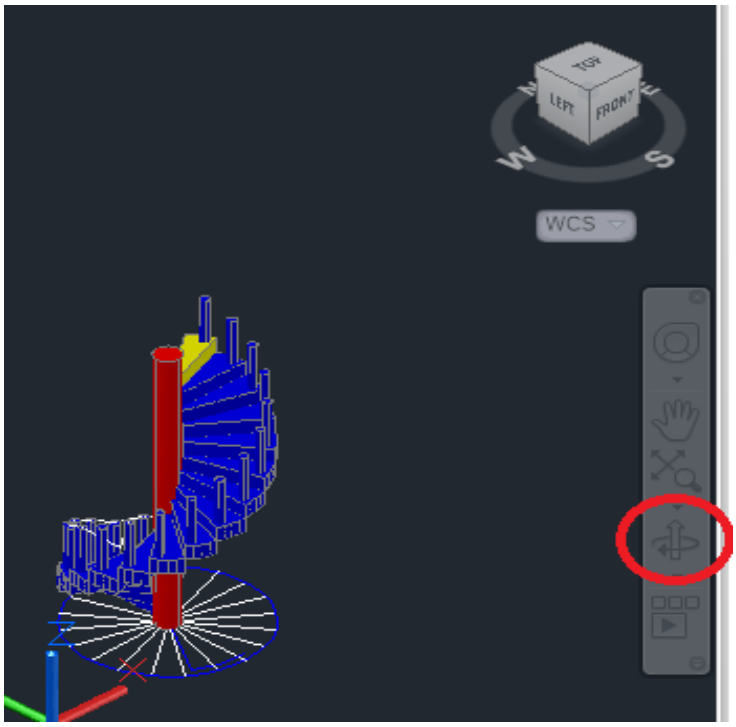
3D Orbit is used to view the selected object/s (Selection must be apply before the command of 3D Orbit) or the entire drawing within the current viewport from any angle.

When 3DOrbit is active, the target of the view stays stationary and the camera location, or point of view, moves around the target. However, from the user's point of view, it appears as if the 3D model is turning as the mouse cursor is dragged. In this way, you can specify any view of the model.

You can activate the 3D Orbit by the following methods:

View Tab	Button	Command Entry
Orbit		3dorbit





There are nine types of Orbiting, which may be used during orbiting.

You can get the first three orbits from **3D Modelling→View→Navigate**

While other will you get from other Navigation option from shortcut menu, during the orbiting process as you will see later in this unit.

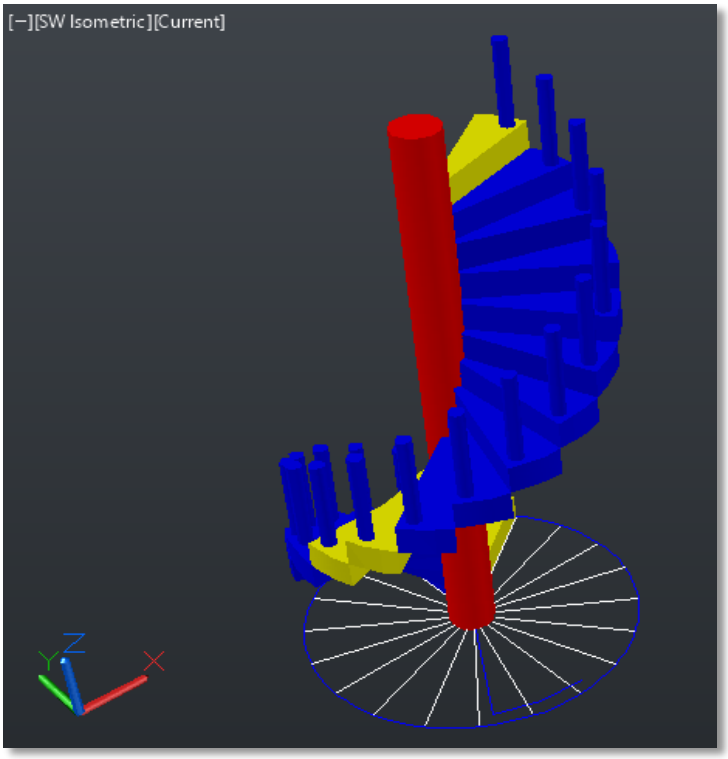
Sr#	Name	Command	Description
1	Orbit	Orbit/3dorbit	Constrains orbiting to the XY plane or the Z direction.
2	Free Orbit	3dforbit	Allows orbiting in any direction, without being constrained to the XY plane or the Z direction.
3	Continuous Orbit	3dcorbit	Changes the cursor to a sphere with two continuous lines encircling it and enables you to set the objects into continuous motion.
4	Adjust Distance	3DDISTANCE	Simulates moving the camera closer to the object or farther away.
5	Walk	3DWALK	Changes the cursor to a plus sign and enables you to "walk through" a model at a fixed height above the XY plane, by dynamically controlling the location and target of the camera. Walk orbit works only in perspective view
6	Swivel	3DSWIVEL	Changes the cursor to an arched arrow and simulates the effect of swiveling the camera.
7	Fly	3DFLY.	Changes the cursor to a plus sign and enables you to "fly through" a model without being restricted to a fixed height above the XY plane. Fly orbit works only in perspective view
8	Zoom	3DZOOM	Changes the cursor to a magnifying glass with plus (+) and minus (-) sign and simulates moving the camera closer to an object or further away. Works like the Adjust Distance option.
9	Pan	3DPAN	Changes the cursor to a hand cursor and moves the view in the direction that you drag.

Procedure
Apply Orbiting to Drawing

STEP BY STEP Operations on AutoCAD

GET READY. Before you going to begin these steps, make sure to set unit and you must have objects in your drawing.

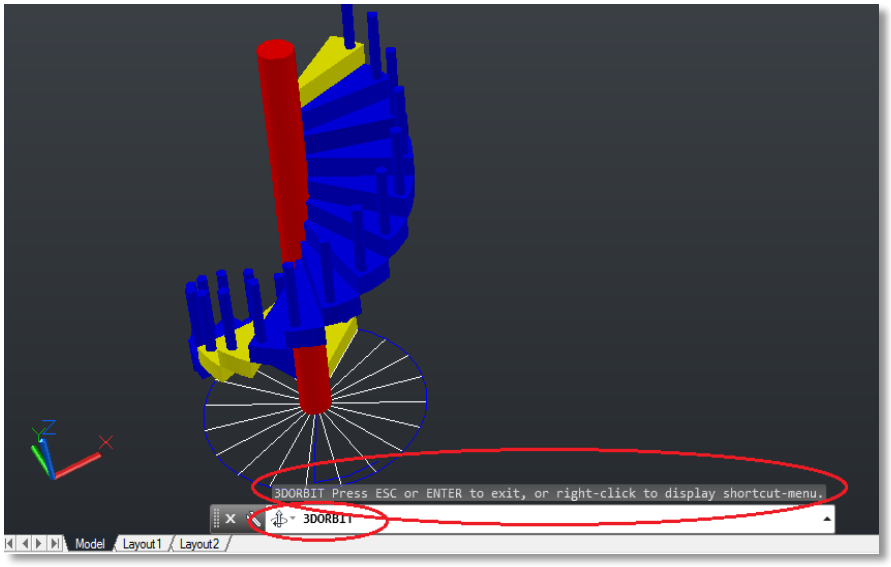
We are going to apply 3D Orbit on the following drawing as shown in the figure below



Step-1: Type **orbit/3dorbit** command in command bar <Enter>

Step-2: You can orbit/revolve the drawing according to XY plane or the Z direction

During any orbiting you can get other functions as shown below:



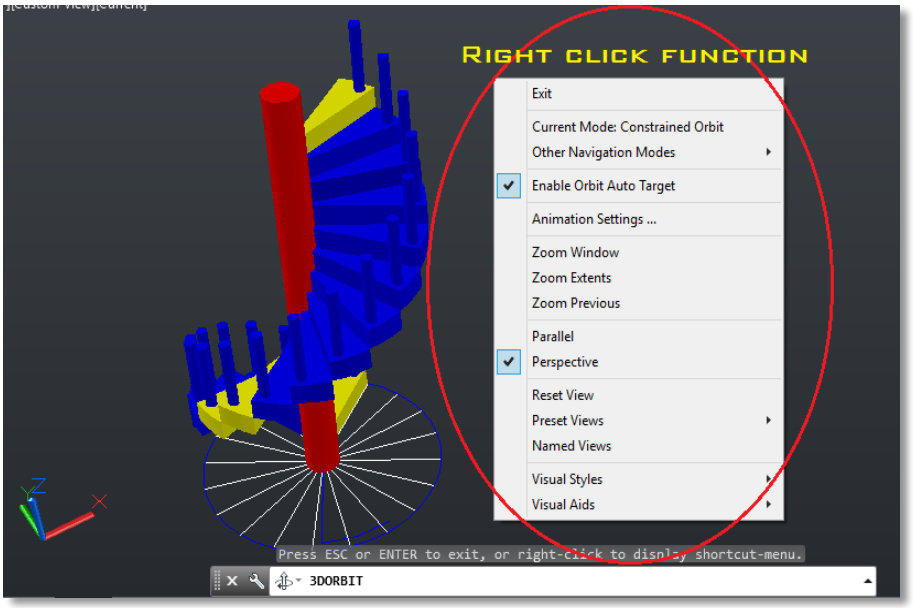
Whenever you apply any orbit, you will get the option “Press **ECS or ENTER** to exit or **right-click** to display shortcut-menu”.

2. Use of ECS Button

When you press ECS button during orbiting, as the name describe ECSAPE, you will get release or exit from the orbit command, and will get ready to prompt other commands of AutoCAD.

3. Use of Right Click

Whenever you right-click during orbiting you will get the following shortcut-menu as shown in the figure below

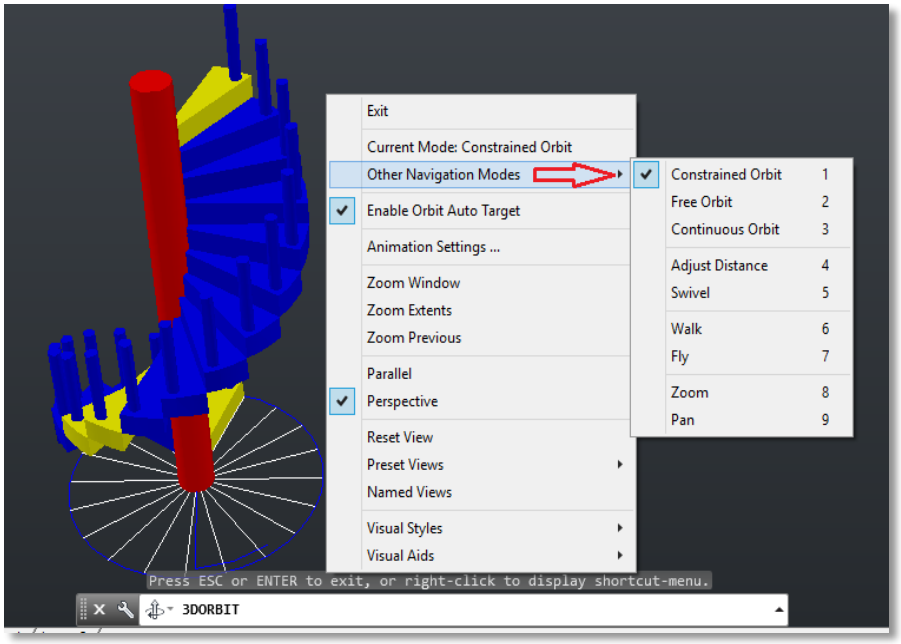


4. Discover Other Navigational Modes

As we discussed other navigation from where you can get the nine orbiting types as shown in the figure below.

Step-1: Type **orbit/3dorbit** command in command bar <Enter>

Step-2: Right click at the drawing area, you will get the following



5. 3D Orbit on Selected Object

3D Orbit on selected object/s allows you to orbit / revolve the specific object within the drawing.

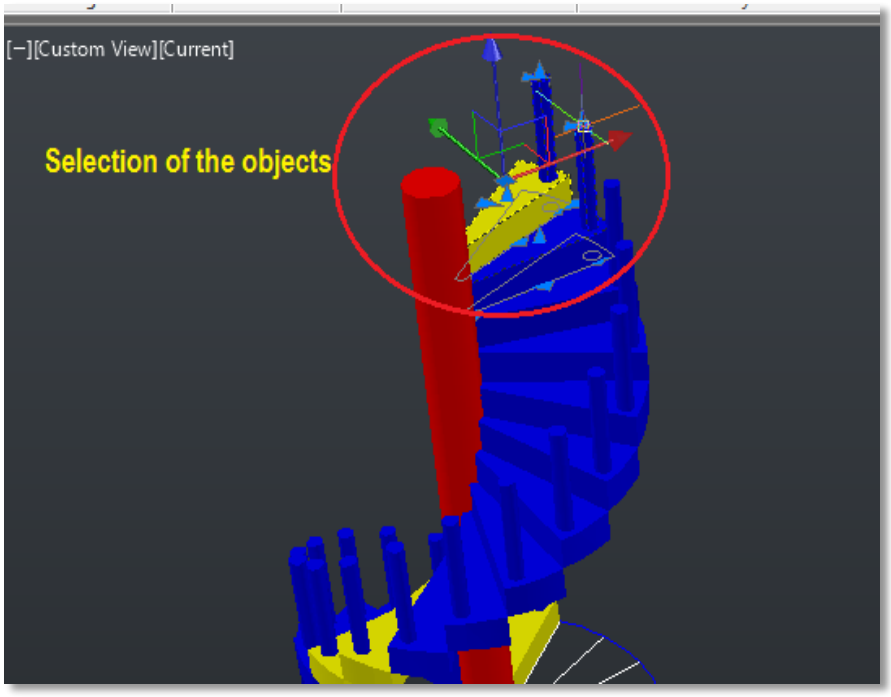
Procedure
Apply Orbiting to selected object

STEP BY STEP Operations on AutoCAD

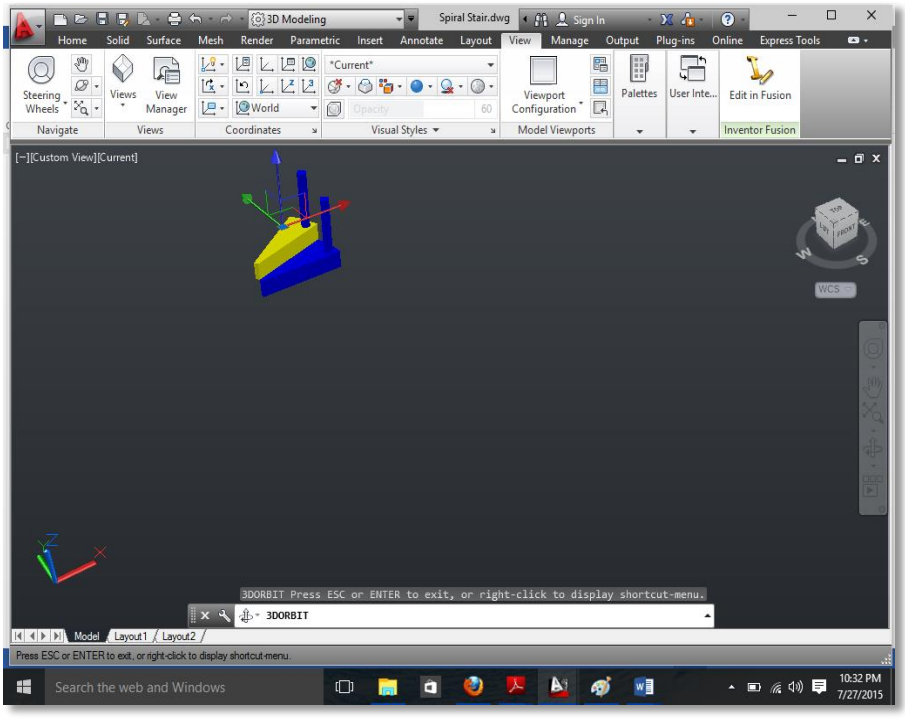
GET READY. Before you begin these steps, be sure to set unit and you must have objects in your drawing.

We are going to apply 3D Orbit on the following selected object within the drawing as shown in the figure below

Step-1: Select the object before applying the orbit command, as shown here



Step-2: Type **orbit/3dorbit** command in command bar <Enter> you will get the only selected object to orbit as shown in the figure below



Step-3: You can orbit/revolve the drawing according to XY plane or the Z direction.

Step-4: When you press ECS button, you will get display the whole drawing again.

6. Developing Pan and Zoom in 3D Orbit

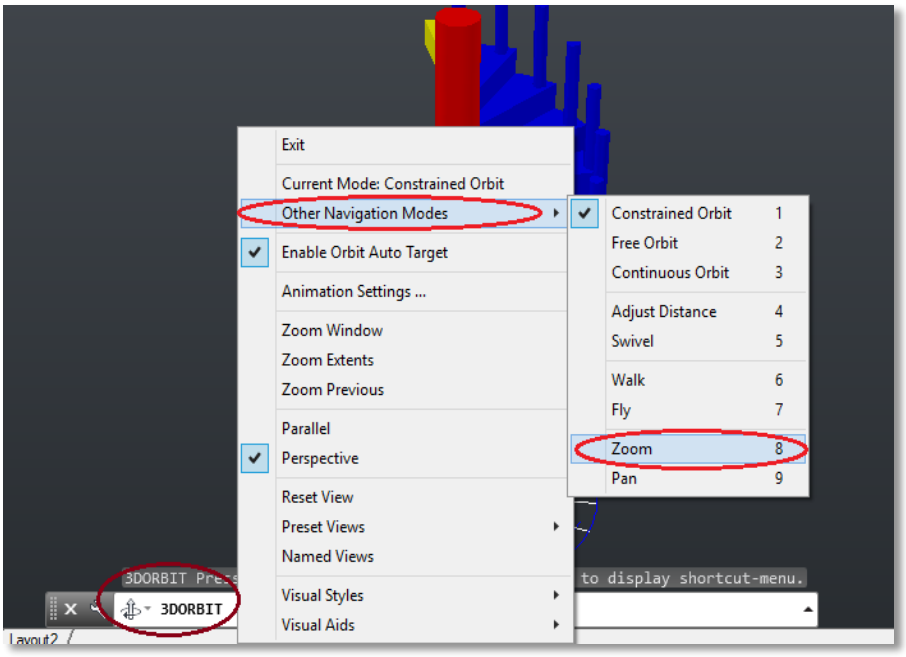
You can easily develop Zoom and Pan command during orbiting, the function of zoom and pan as the same as we discussed in previous module, but the only difference is to get Zoom and Pan after applying the orbit command then orbit/revolve that area.

Procedure
Developing Pan and Zoom in 3D Orbit

Step-1: Type **orbit** or any other **orbit** command in command bar <Enter>

Step-2: Right click at the drawing area.

Step-3: Go through Other Navigational Mode→**zoom** as shown in figure below



Step-4: **Zoom**-In or **Zoom-Out** the object or drawing.

Step-5: Then Right click again at the drawing area.

Step-6: Go through Other Navigational Mode→**constrained Orbit** and continuing to orbiting as you need.

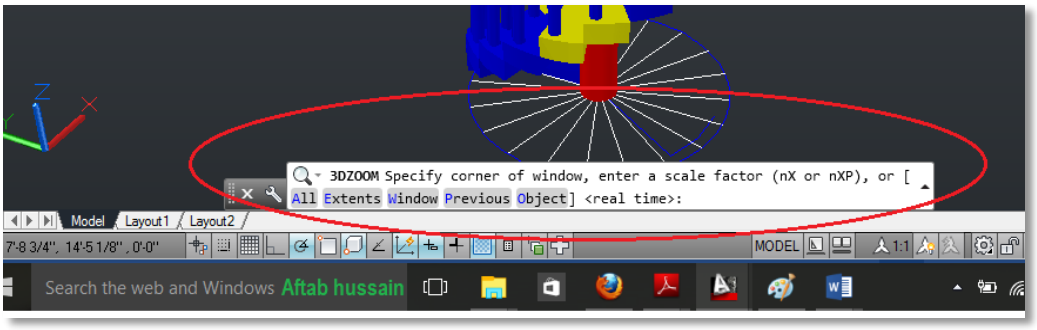
Step-7: Repeat step-1 to step-6 for Pan and get the Pan functions.

Alternate Method for 3D Orbit Zoom

You can also get all zooming option (extent, window, previous etc.) During orbiting by using direct command.

Command to Enter: **3DZOOM**

Step-1: Type **3dzoom** command in command bar <Enter>, you will get the following



Remember

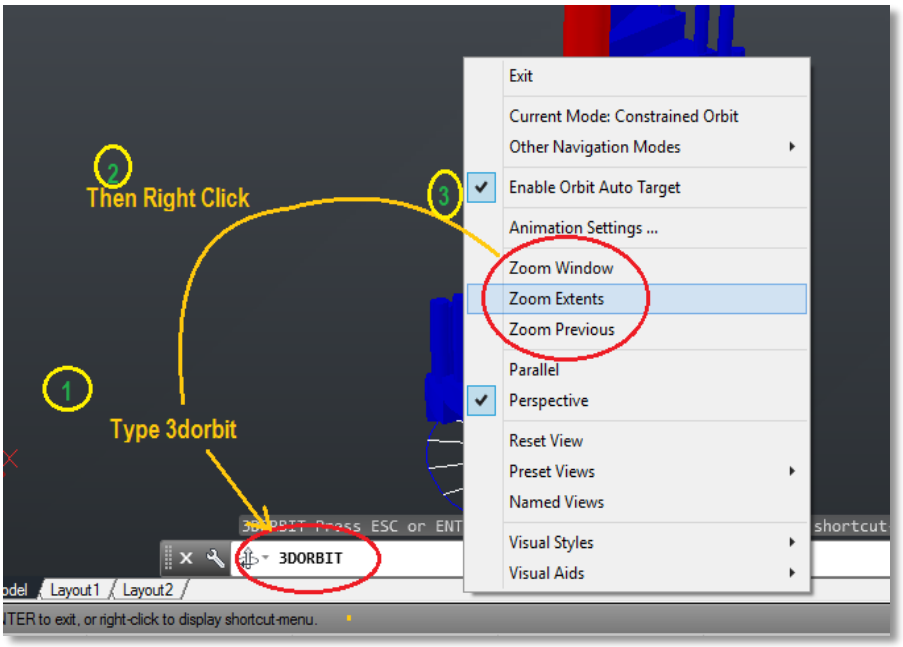
All Zooming option work the same as we discussed in Previous Modules, if you need just glance back to the previous modules.

Step-2: From here you can apply any of the zoom command as shown in the options to orbit the drawing or object/s.

Learner's Activity

Get through the following procedure and apply the three Zooming (Extents, Window and Previous) Options on any drawing or object.

Procedure



7. Projection mode by selecting “Perspective” option in 3D Orbit

Before applying projection mode we must know about the Perspective View and Parallel view

Difference between Perspective and Parallel Projection

Parallel Projection

In the parallel view the objects are display in two parallel lines and that line in a drawing never converge. The shapes never change and never distorted in the drawing when they are closer.

Perspective Projection

In the perspective projection the object are seeing larger when they’re up close and smaller at a distance and all parallel lines converge at one point. The shapes are change when the objects are very close unlike parallel projection.

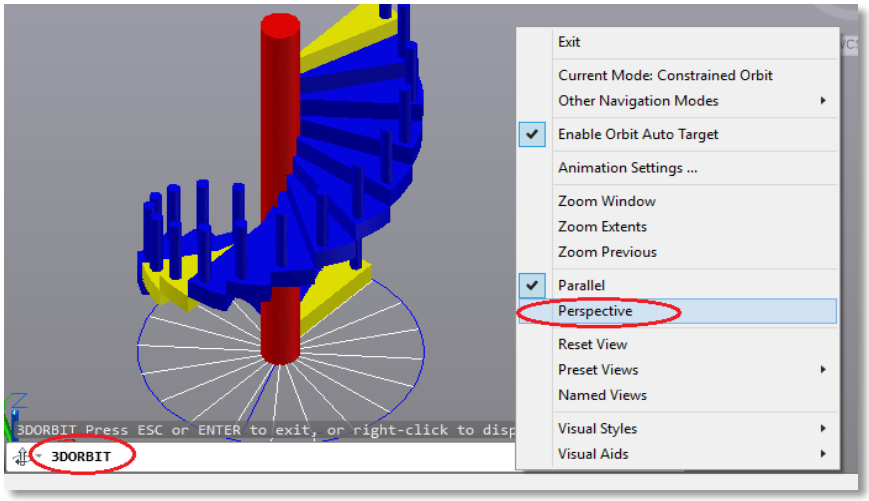
Procedure
Apply Projection Mode

Projection Mode: We will apply projection mode by selecting “Perspective” option in 3D orbit through **Fly** and **walk** option of the orbit.

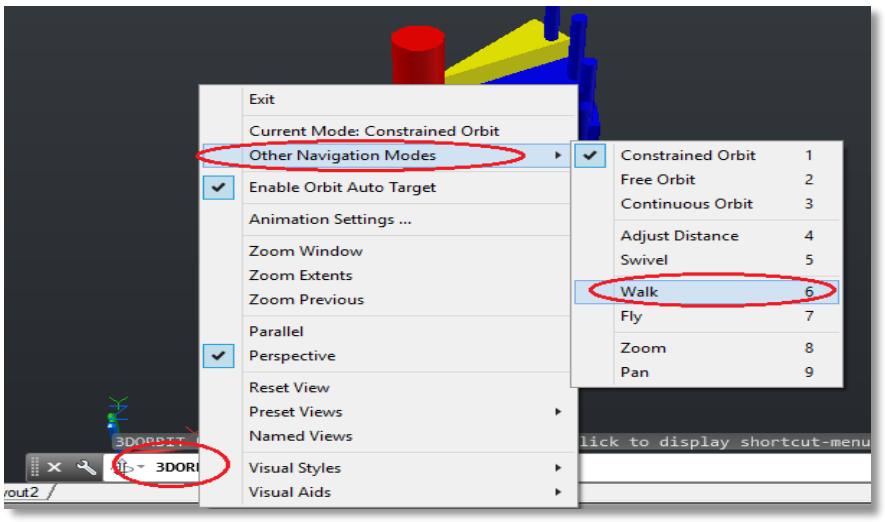
Note:
Fly and Walk orbit works only in perspective projection.

Step-1: Type **3dorbit** command in command bar **<Enter>**

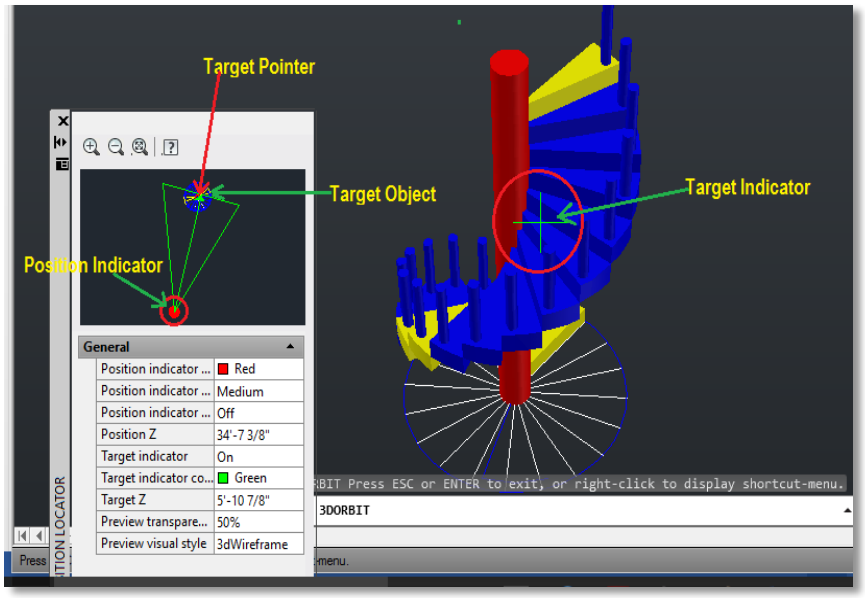
Step-2: Right click in the drawing area, and change the projection into Perspective as shown in the figure below



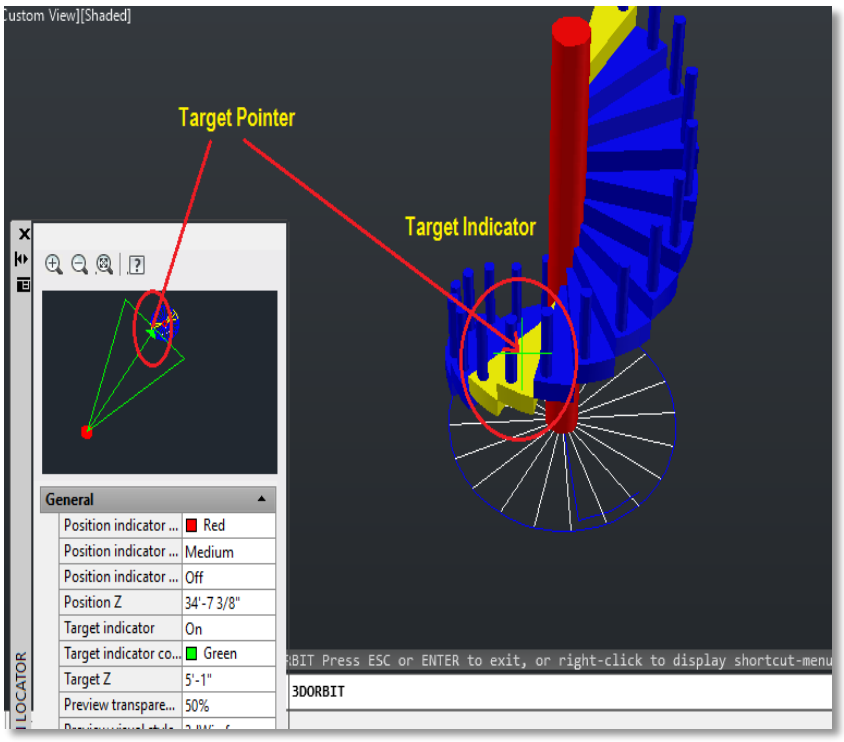
Step-3: Right click again and go through other navigation to select walk option as shown in the figure below:



You will get the following perspective projection as shown in the figure below:



Step-4 You can move the position indicator, Target Pointer and Object to orbit the object in different perspective as shown in figure below



Target Pointer indicated the target indicator and you can view the yellow shaded step easily.

Learner's Activity

Repeat the above steps for the fly Projection.

8. Different Visual Styles in 3D Orbit

In previous module we already discussed Visual Styles in detail but here we are going to get practice on Visual styles in 3D Orbit perspective.

Visual styles composites of the following styles in 3D Orbit.

1) 2D Wireframe
2) Conceptual
3) Hidden
4) Realistic
5) Shaded
6) Shaded with edges
7) Shaded of Gray
8) Sketch of Gray
9) Wireframe
10) X-ray

Procedure

Apply Visual Styles in 3D Orbit

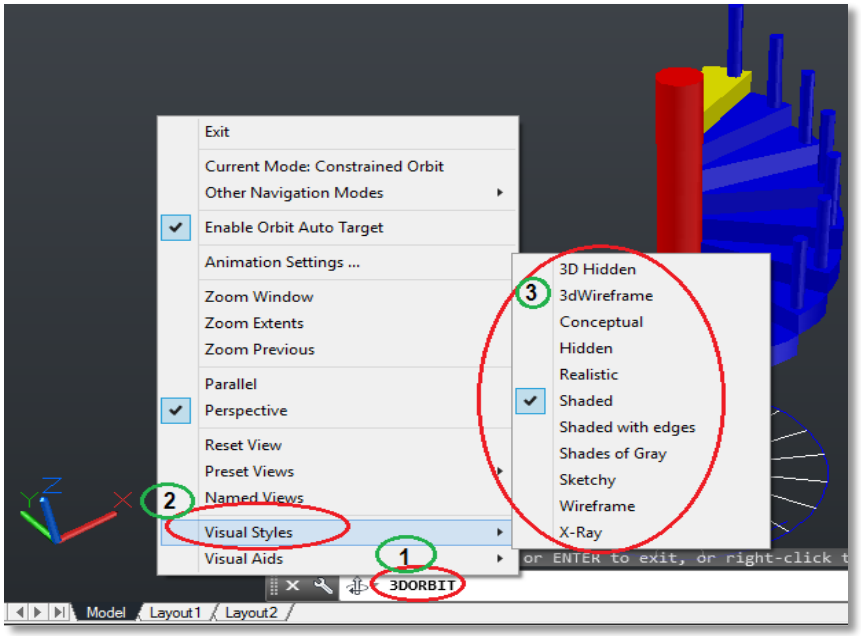
STEP BY STEP Operations on AutoCAD

GET READY. Before you begin these steps, be sure to set unit and you must have an object in your drawing to apply visual styles on it.

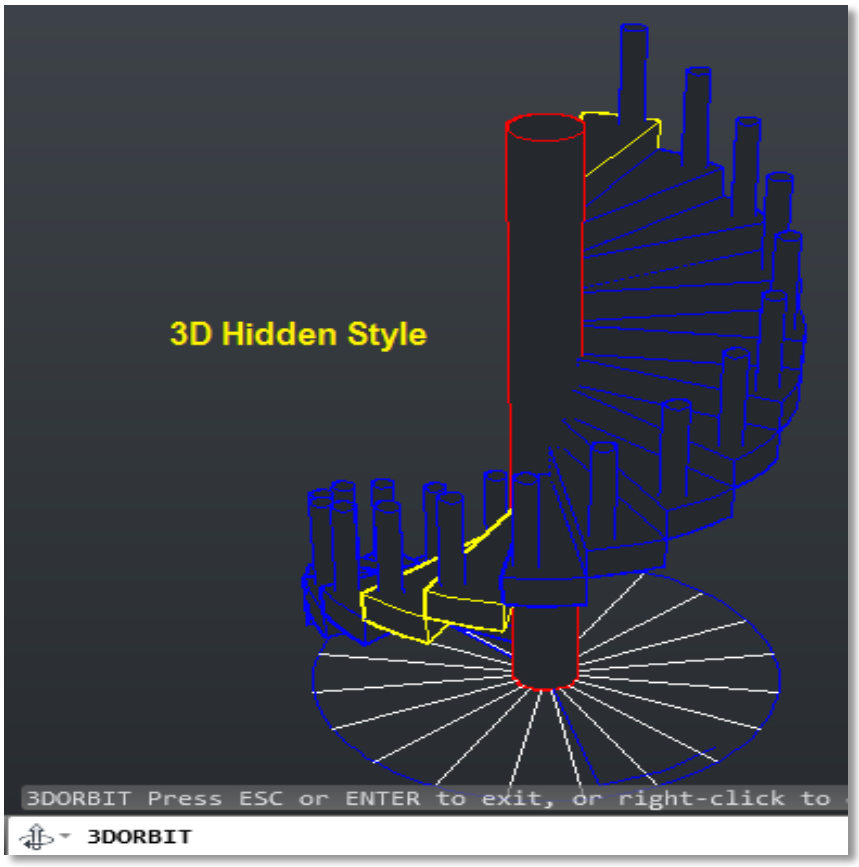
Step-1: Type **3dorbit** command in command bar **<Enter>**

Step-2: Right click in the drawing area.

Step-3: Go through **Visual styles** from the Popup menu as shown in the figure below.

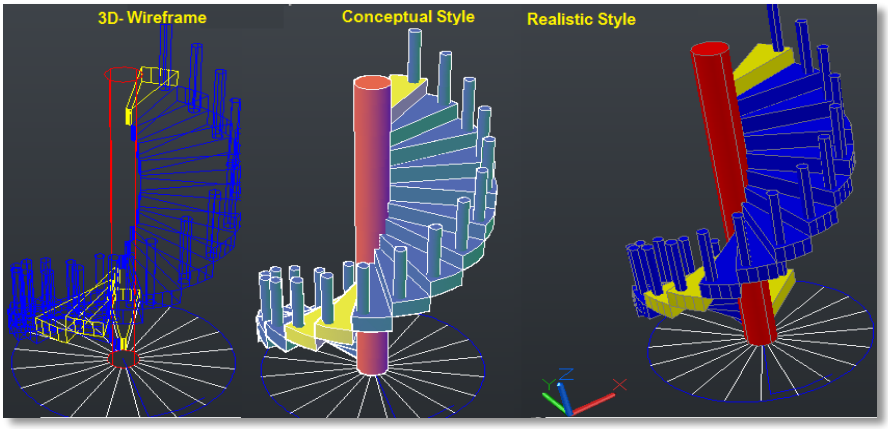


Step-4: Select **3D Hidden** Styles, You will get the following figure below



Step-5: Repeat **Step-4** for getting different visual styles in 3D Orbits.

The following figures shown different visual styles:



9. Different Visual Aids in 3D Orbit

Visual Aids provides views to visualize the objects.

There are three different visual aids in 3D Orbit

1) **Compass:** Through Compass its draws a 3D sphere composed of three coordinates representing the *X*, *Y*, and *Z* axes.
2) **Grid:** Grid is used to display the grids line which is oriented along the *X* and *Y* axes. Grid is already discussed in module-2
3) **UCS Icon**: UCS Icon is used to show or hide the UCS Icon as discussed in previous module.

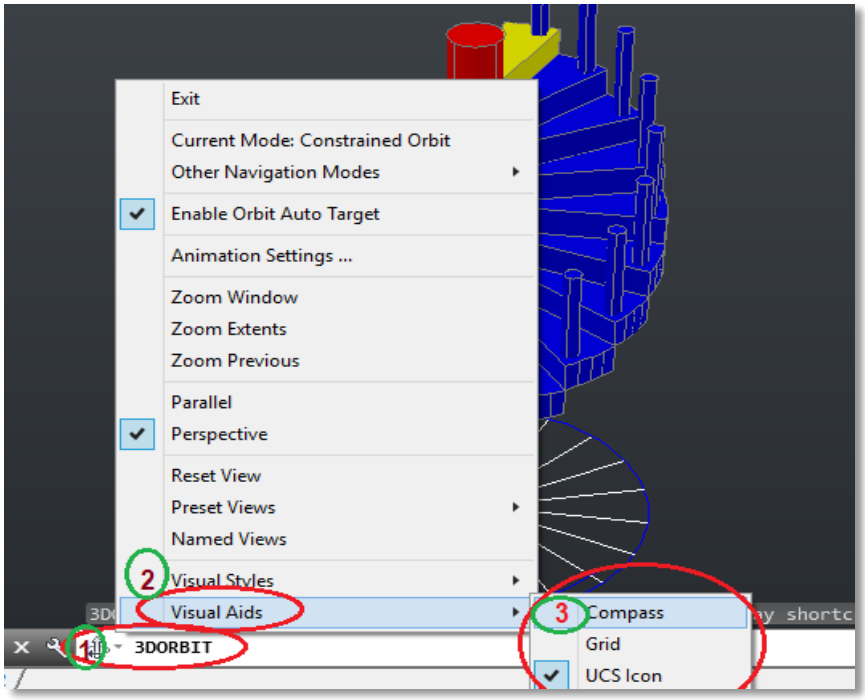
Procedure

Apply Visual Aids in 3D Orbit

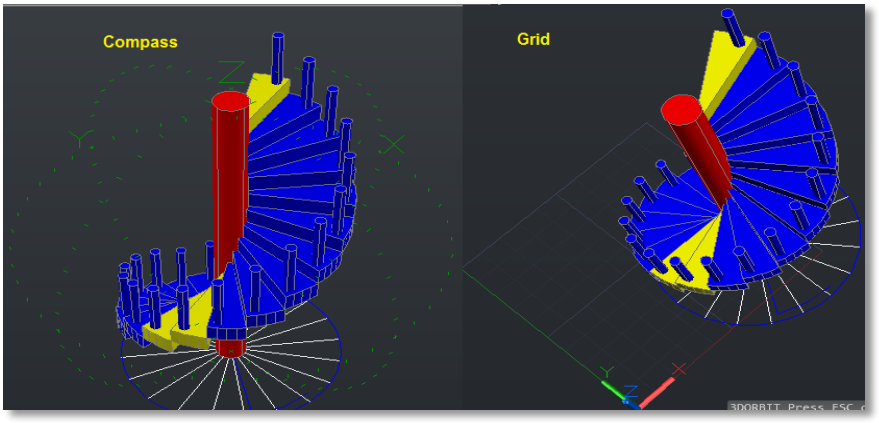
Step-1: Type **3dorbit** command in command bar **<Enter>**

Step-2: Right click in the drawing area.

Step-3: Go through **Visual aids** from the Popup menu as shown in the figure below



Step-4: Select Visual Aids option one by one you will get the following Aids as shown in the figure below.



10. Preset views in 3D Orbit

Preset View: Displays a list of predefined views such as Top, Bottom, Front, Back, Left, Right and SW, SE, NW, NE Isometrics. Choose a view from the list to change the current view of your model.

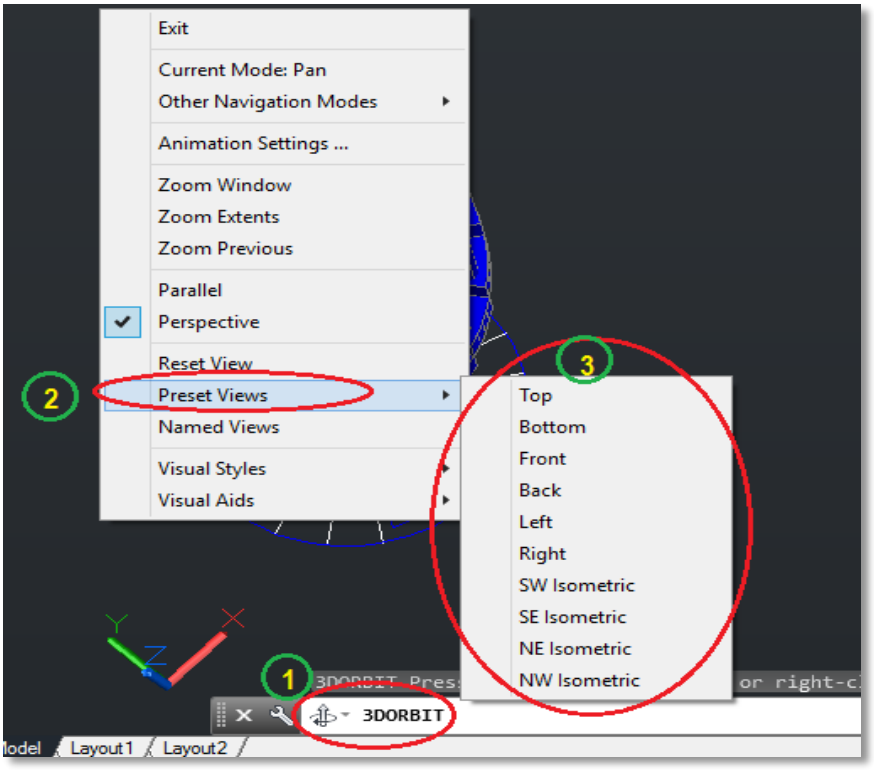
Procedure

Apply Visual Aids in 3D Orbit

Step-1: Type **3dorbit** command in command bar **<Enter>**

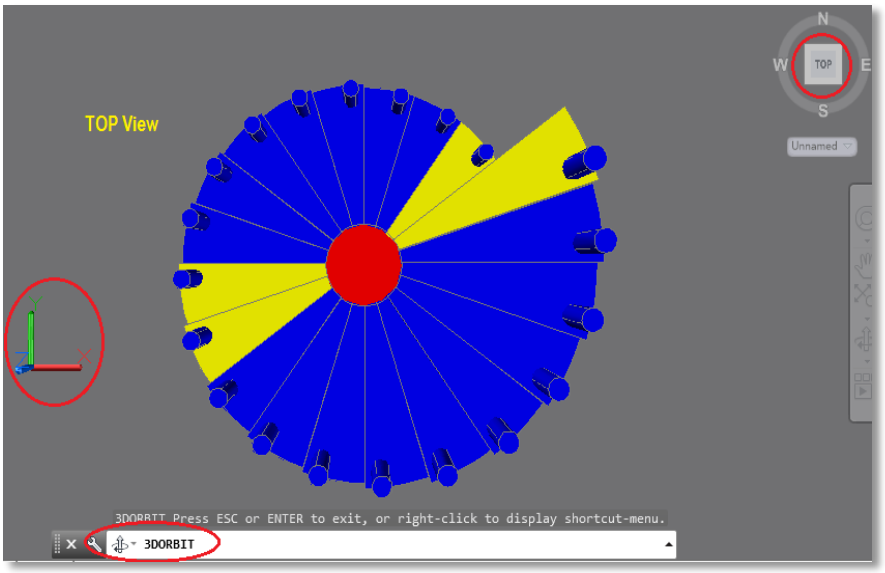
Step-2: Right click in the drawing area.

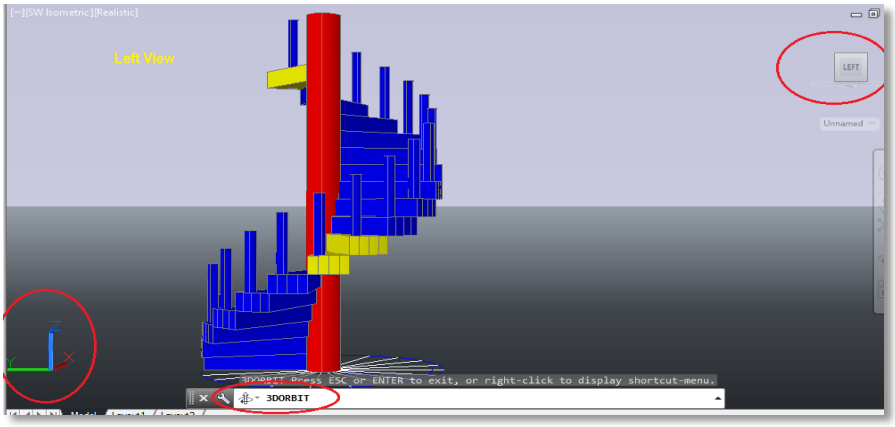
Step-3: Go through **Preset View** from the Popup menu as shown in the figure below:

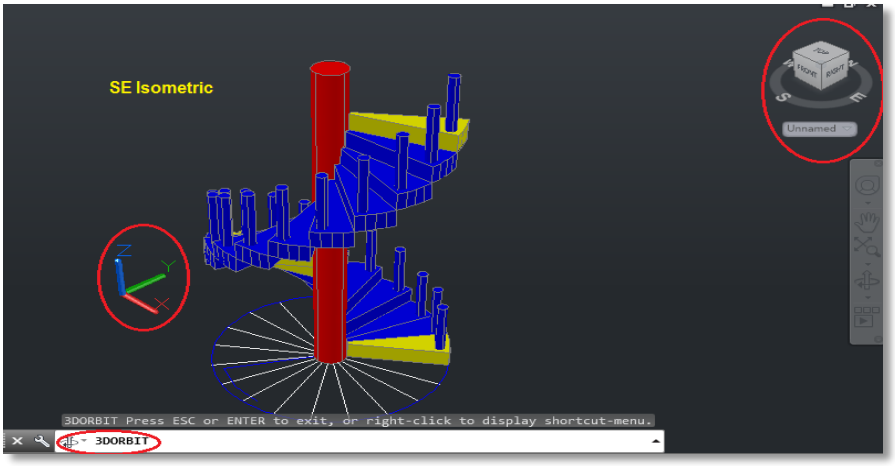


Step-4: You can Apply Present view one by once as shown in figure above.

Following are some preset view applied



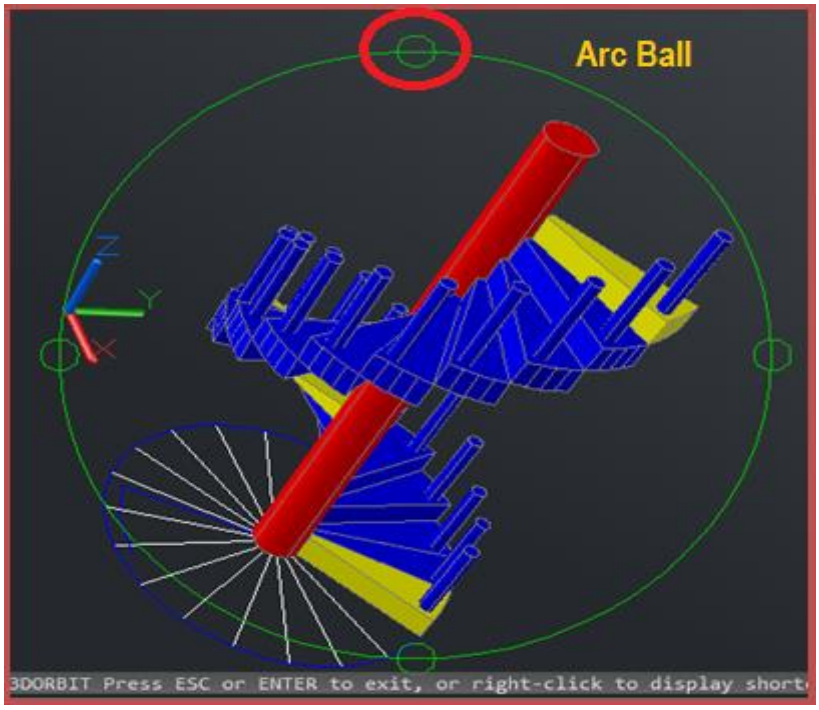




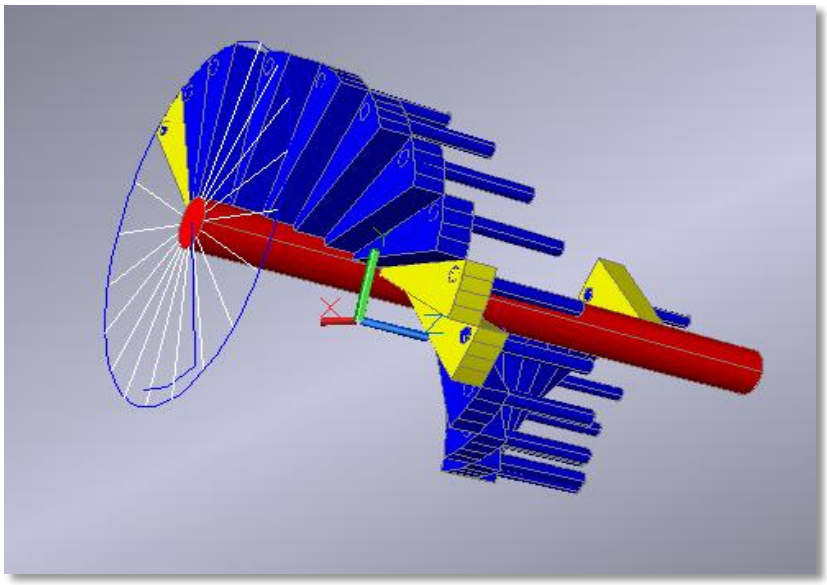
11. Differentiate between Free and Continuous Orbit

Free Obit: Allows orbiting in any direction, without being constrained to the XY plane or the Z direction.

Click on the arc ball to change the view of the object as shown in figure below



Continuous Orbit: Changes the cursor to a sphere with two continuous lines encircling it and enables you to set the objects into continuous motion.



Note: You can stop continuous orbiting by pressing ESC button on the keyboard.

Learning Unit 2: Perform 3D Navigation

Overview

This unit describe familiarity with AutoCAD Camera setting for the objects in the drawing, after completing this unit you will be able to create and design camera in the drawing to view object from a specific sides either with narrow field view or with large field view.

In this unit you will also learn more functions of camera including View, Preview, properties, Plotting, Display, distance etc. to define parallel projection or perspective views by using a camera and target with the help of **"DVIEW"** command.

Further you will be able to simulate walking and flying through 3D faces to travel along the XY plane and apply **"ANIPATH"** command for motion path animations, such as 3D animated walk-throughs of a model, allow you to visually demonstrate a model to both a technical and non-technical audience. You can record and play back a navigation to communicate your design intent dynamically.

1. Creating Camera

Camera is used to define a 3D view, you can place camera anywhere in the drawing to define a specific 3D view.

You can Set a camera and target location to create and save 3D perspective views of objects. Further you can defining its name, height, lens length, and clipping planes.

To Access Camera

Render Tab	Button	Command Entry
Camera panel →Create Camera		Cam/camera

Procedure

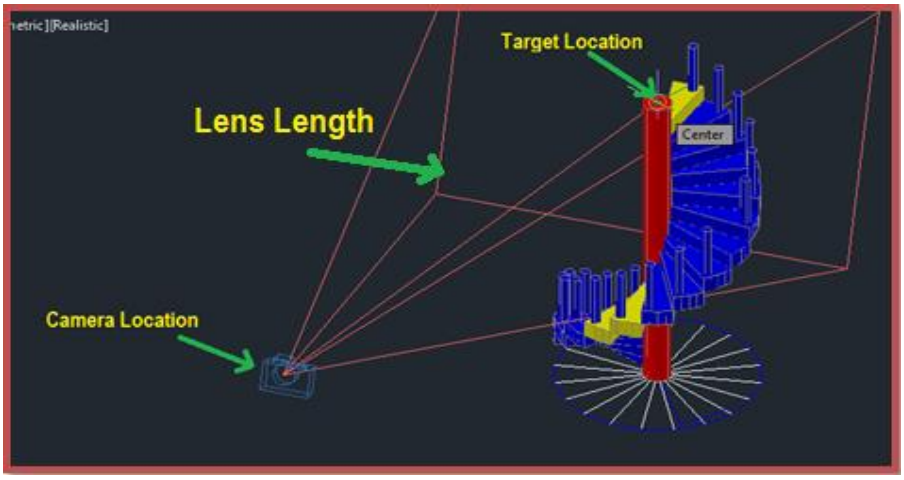
Apply Visual Styles in 3D Orbit

Step-1: Type **cam/camera** command in command bar **<Enter>**

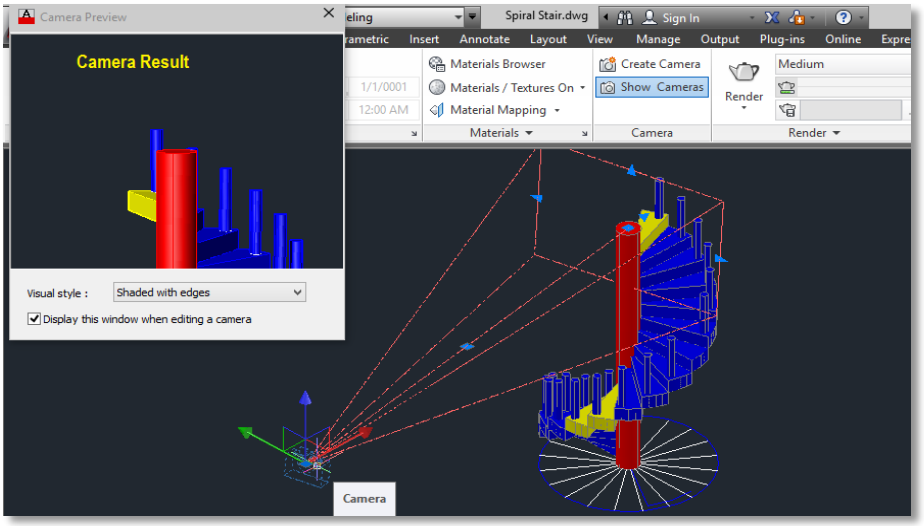
Step-2: Click in the drawing to specify the camera's location.

Step-3: Click again in the drawing to specify a target location.

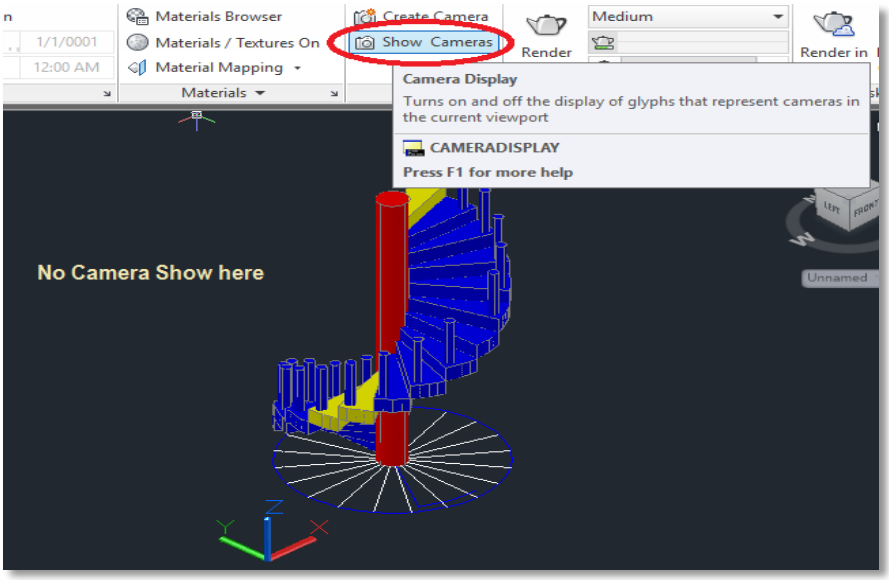
Step-4: Press Enter to Finish, you will get the following as shown in figure below



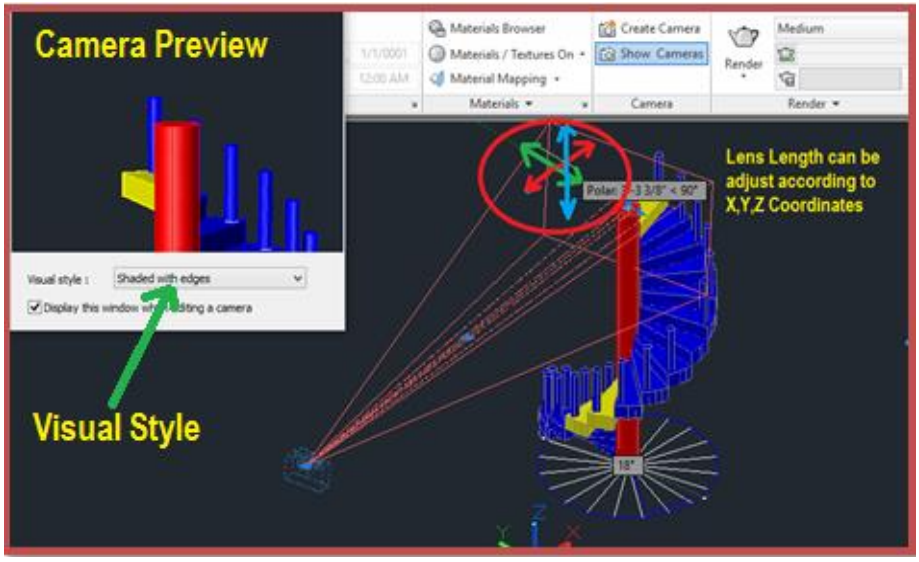
Step-4: To View the Camera, just click the camera location, you will get the following as shown in the figure below



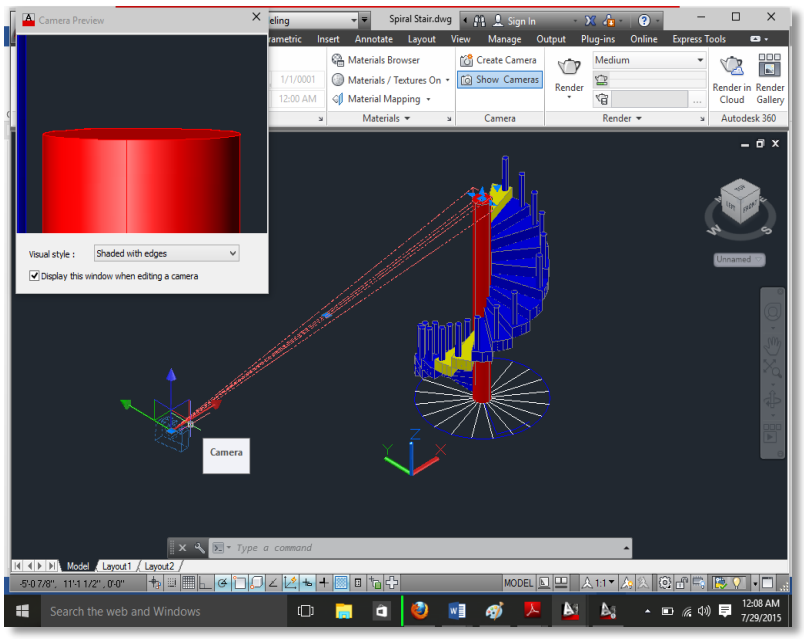
Step-5: You can show/hide the camera by clicking show camera button as shown in figure below.



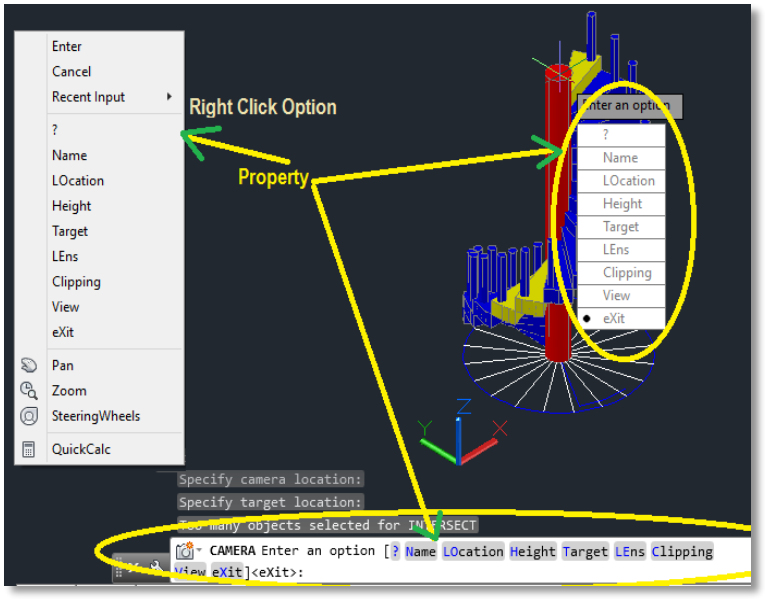
Step-6: Lens length setting can be adjust according to X, Y and Z coordinates as shown in the figure below.



- Click and drag grips to resize or relocate a lens length or field of view.
- Narrow the focus of lens on the target greater the field view and greater the lens length, the narrower the field of view.
- You can change the visual style of the view through camera preview option.



- To further define camera properties, you will get properties on the screen while you setting up camera or you can right-click and select from the list of options.



2. Camera Properties

? Display a list of the currently defined Cameras if any.

Just enter the name of the Camera or press Enter to list all cameras if created.

Name You can Enter Name of the New Camera, if you never enter name of the camera, by default it will named camera1, camera2… so on.

Location Specifies the location of the camera as we discussed previously.

Height You can define or changes the camera height.

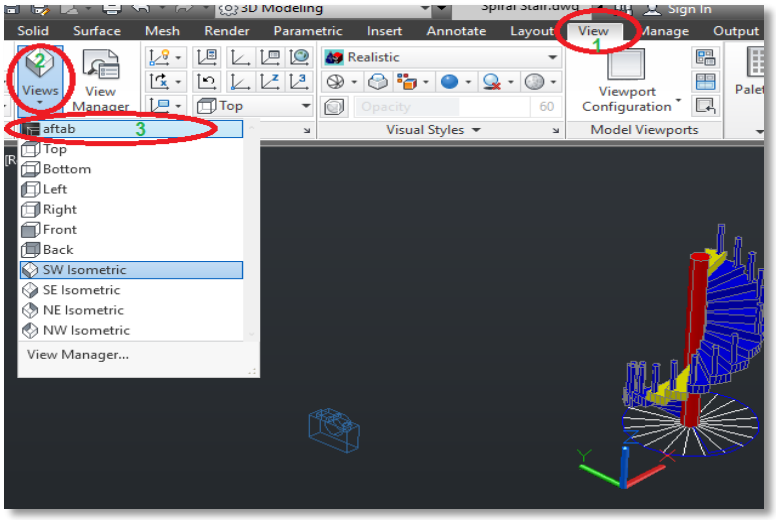
Target Specifies the target of the camera as we discussed previously.

Lens Changes the lens length of the camera as we discussed previously.

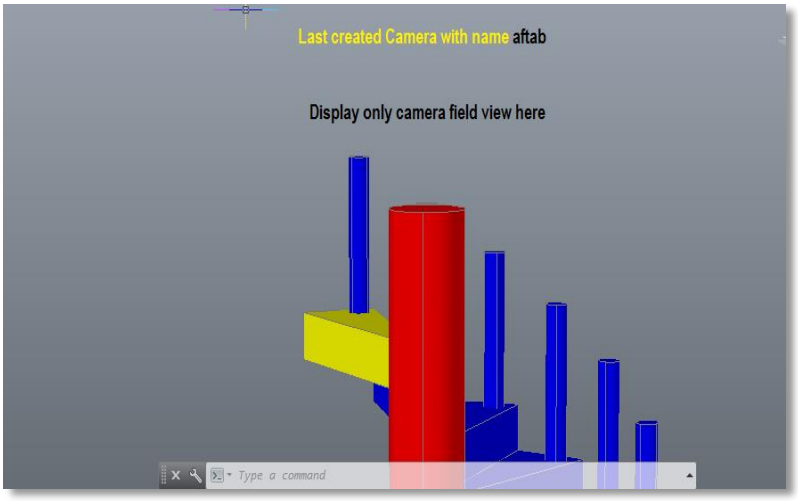
Clipping Defines front and back clipping planes and sets their values.

3. View Camera and Its Properties

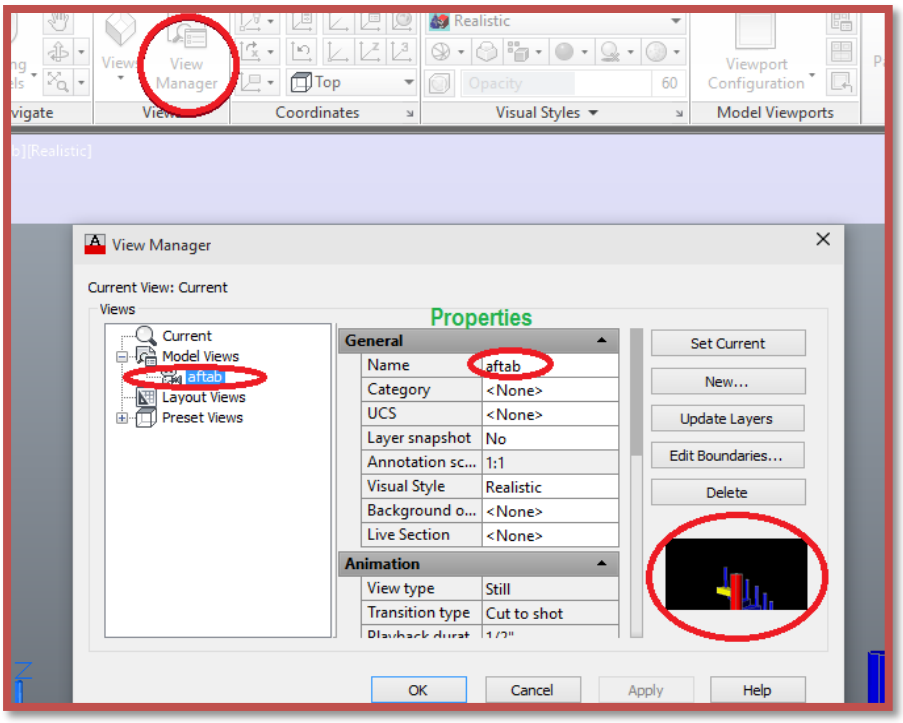
If you created any camera view and Named that camera then you can get you view by its camera name from the View Tab→Views option as shown in the figure below.



- Click on the camera name and get the last created camera view as shown in the figure below.



- You can also use View Manager to make changes in the camera view properties.



4. Plotting and Swivel a Camera

So simple just double click on camera and get a quick properties of a camera

- Click **Yes** under the Plot option to plot a camera.
- Enter **3DSWIVEL** command and simulates panning with a camera in the direction that you drag. The target of the view changes. You can swivel the view along the XY plane or along the Z axis.

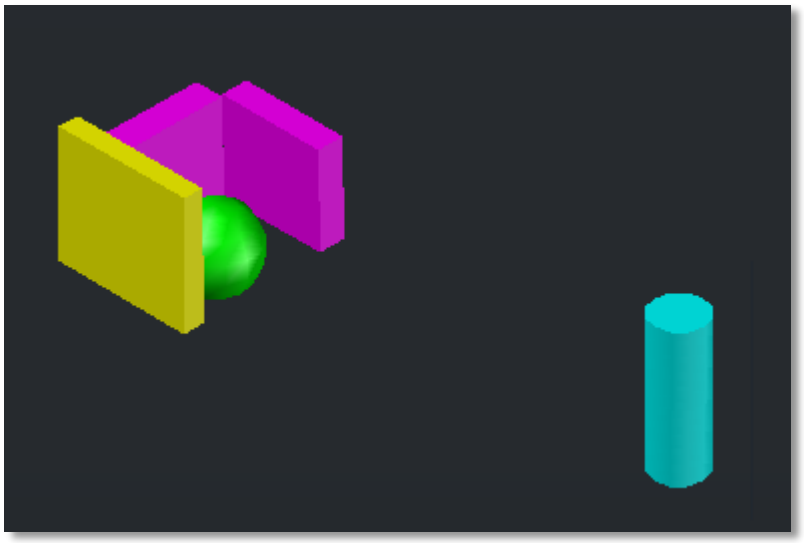
5. DVIEW

Dynamic VIEW defines parallel projection or perspective views by using a camera and target. **DVIEW** that gives you more control over the view of your object.

Procedure

DVIEW Command

Appling Dynamic view for Perspective view to the following object



Step-1: Change the view into perspective view by using

- **Perspective** Command
- Enter new value for PERSPECTIVE <0>: 1 {1 = On , 0 = Off)

Step-2: Type **DVIEW** command in command bar **<Enter>**

Step-3: Select objects or <use DVIEWBLOCK>: 1 found

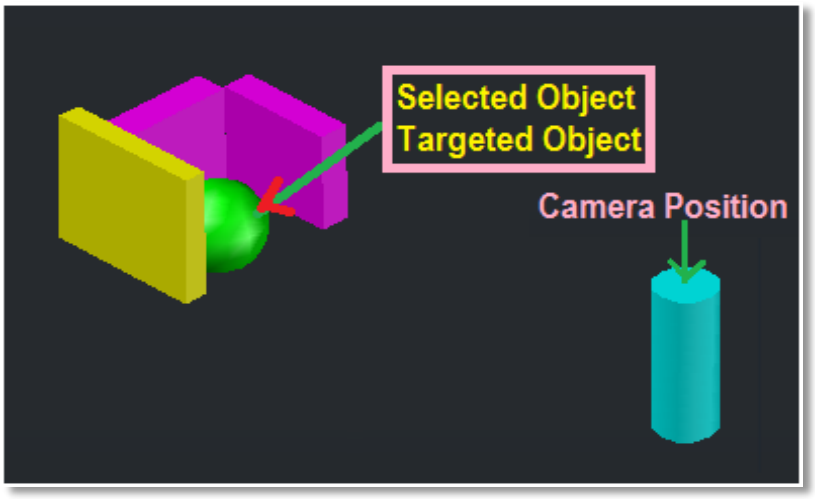
Step-4: Enter option

[CAmera/TArget/Distance/POints/PAn/Zoom/TWist/CLip/Hide/Off/Undo]:

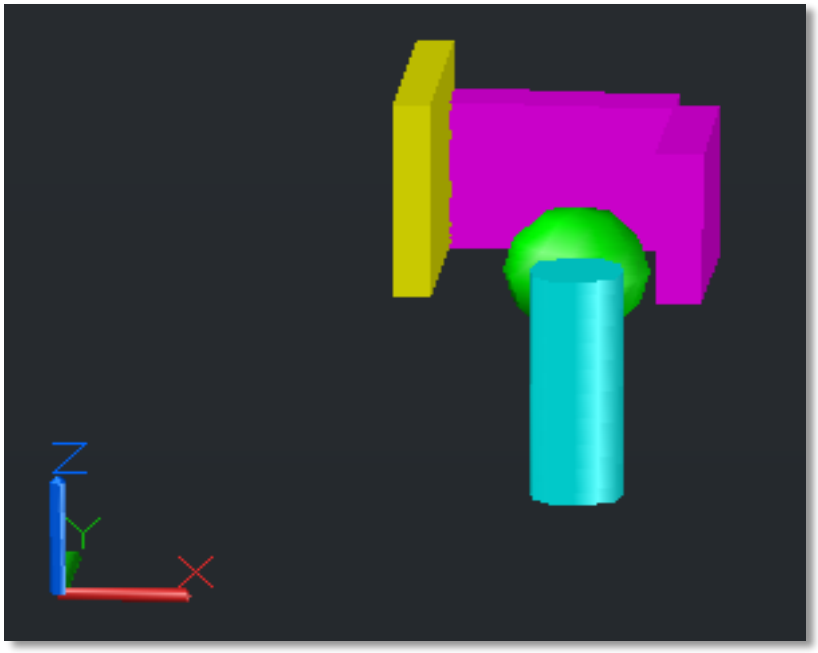
Po

Step-5: Specify target point **<Show in Figure below>**

Step-6: Specify camera point **<Show in Figure below>**



Output of the perspective view under Dynamic view should be as under.



Learner's Activity

You should try to apply DVIEW for the Parallel view and the remaining option of the DVIEW.

1. Walking and flying through a 3D

Walk and fly through 3D have already discussed in **unit-1** of this module, if you want to refresh your knowledge just glance back and refresh your memory.

2. ANIPATH

Animation Path is 3D Camera view through which you can view the object/s in as animated motion via a specified path.

In the drawing, you can create a path object for either the camera or the target.

Note: The path you create is not visible in the animation.

A path can be a line, polyline, 3D polyline, arc, elliptical arc, circle, or spline.

Pay attention on box position

To Access Camera

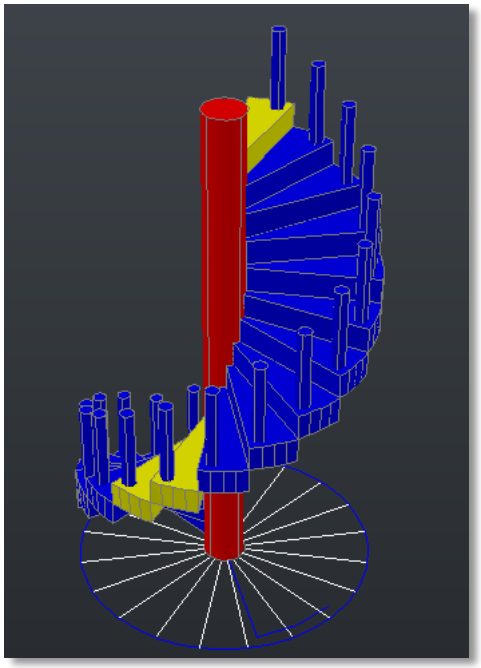
Render Tab	Button	Command Entry
Animation.	Animation Motion Path	ANIPATH

Procedure

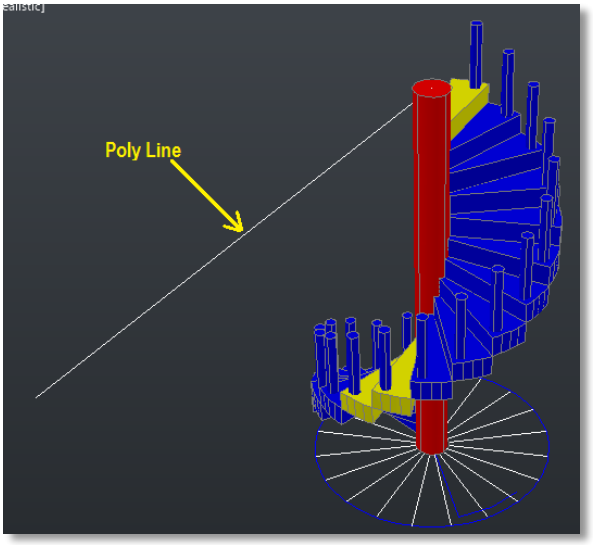
Animation Motion Path

Usually flowing type of Drawings are used through in this Module for practicing purpose, learner can set / draw their own drawing.

We are going to apply animation path on drawing as shown in the figure below.



Step-1: Draw a line/polyline to consider a path for animation.

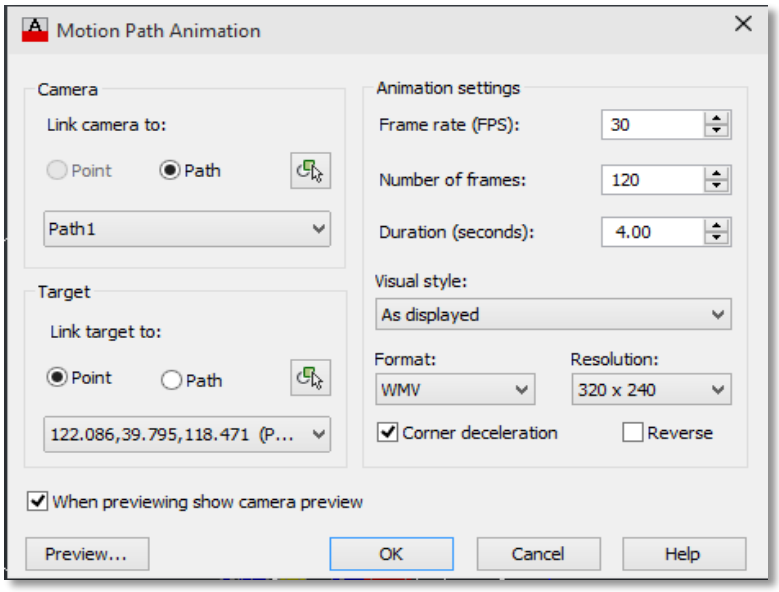


Step-3: Type **ANIPATH** command in command bar or Click Render

Tab→Animations panel →Animation Motion Path.

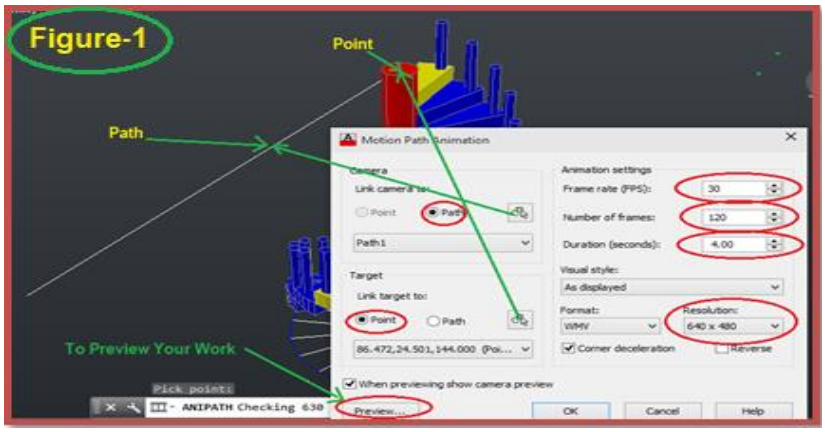
Step-2: In the Motion Path Animations dialog box, do the following:

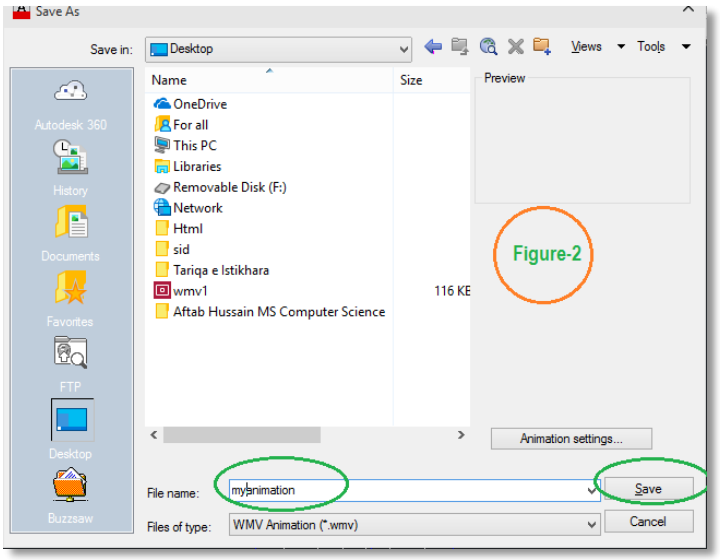
- Specify a point or path for the camera.
- Specify a point or path for the target.
- Modify any animation settings as you want.

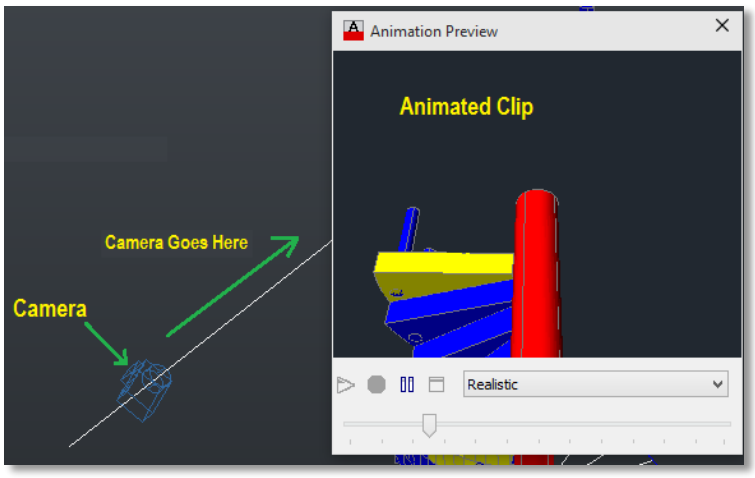


For practice we select.

- Path for the camera.
- Point for the Target.
- Click Preview to View your work.
- Click OK to save the file as shown in the figure-2**.**







Output Look like

Learning Unit 3: Operate 3D Object

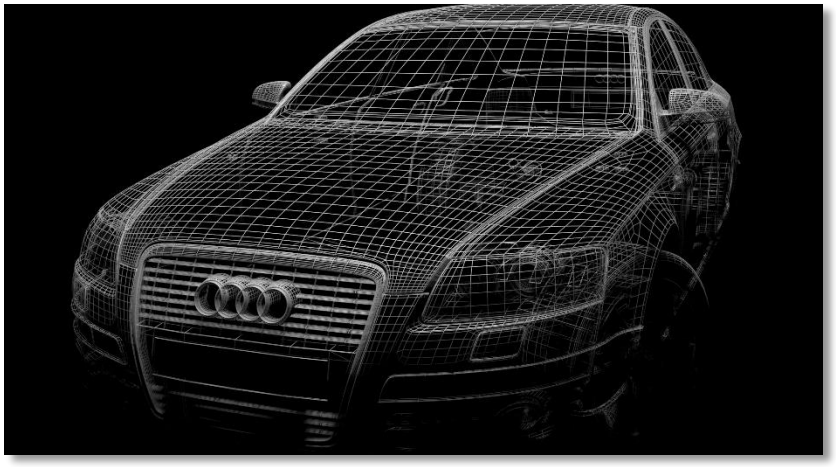
Overview

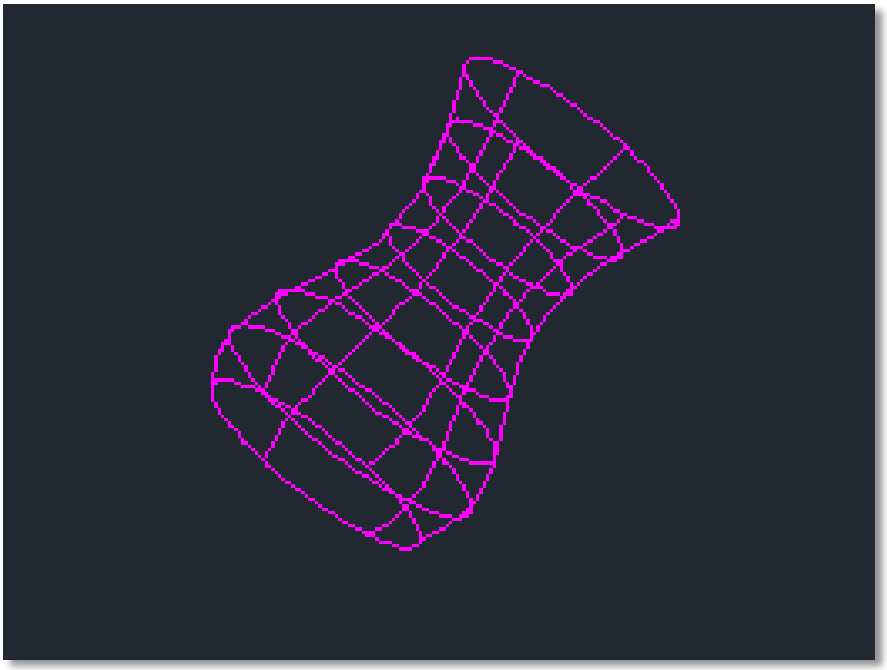
In this unit you will be able to apply surface on different object though different methods and able to create solid by combining different simple shapes of complex solids. After completing this unit, you will able to draw 3D Wire-frame of your 2D and 3D drawing by using lines, curve and edges by applying visual style effect.

1. 3D Wireframe

A **wire-frame model** is a visual presentation of a Two-**dimensional** (2D), three-**dimensional** (3D) or physical object used in **3D computer graphics**. It is created by specifying each edge of the physical object or 3D skeleton by using straight lines or curves.

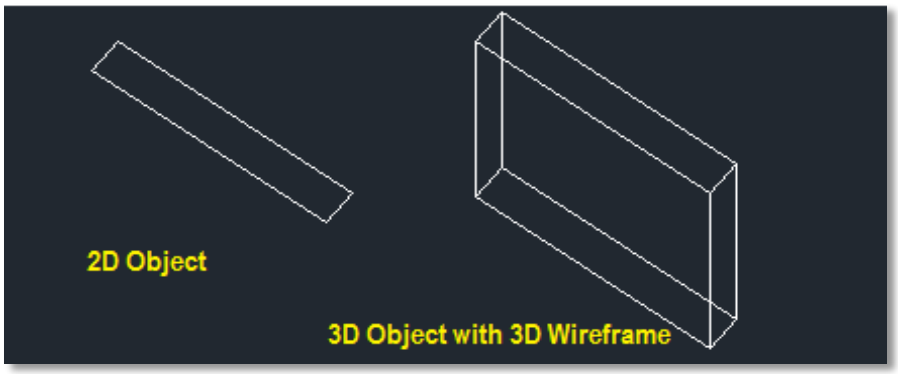
3D Wire frame modeling can be the most time-consuming than 2D drawing.





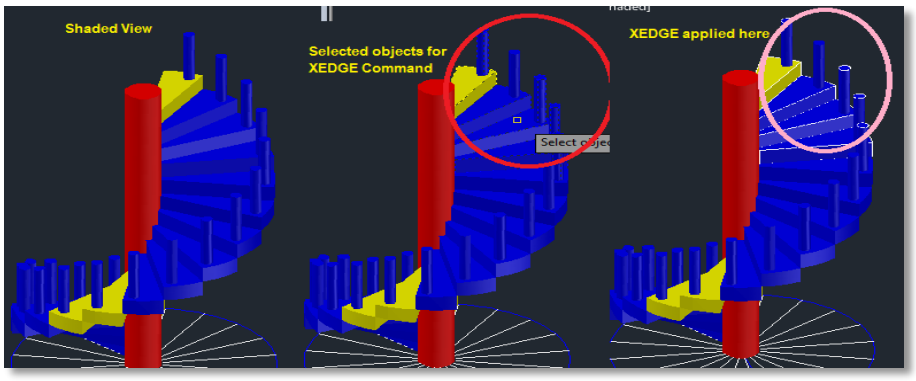
With AutoCAD you can create wireframe models by positioning any 2D planar object anywhere in 3D space by the following method.

Create a 2D object in any view, like draw a rectangle with any dimension, and then convert it in 3D space by using Extrude command as shown in the figure below.



XEDGE Command

XEDGE command is used to create wireframe geometry from regions, 3D solids, surfaces, and meshes as shown in the figure below.



2. Surfaces

We already discussed surface in previous module and will be described in more detail in later modules “Insert surfaces”.

3. Solids

We already discussed surface in previous module and will be described in more detail in later modules “Insert surfaces”.

Solids is the composed of Pre define BOX, CONE , CYLENDER, SPHERE and also can be created by using Extrude , Loft , Presspull , Sweep etc. have already been created in previous module.

Summary of Module

- With the completion of this module we acquainted with the AutoCAD 3D Orbit functionality to rotate the view in 3D space constrained to horizontal and vertical orbit, freely orbiting and continuously orbiting on selected object/s and drawing. We also covered different visual styles, visual aids, preset views and fly and walk view with the collaboration of parallel and perspective projections mode.

- In the second unit of this module we learnt about the camera setting for the objects in the drawing and created different position for the camera in the drawing to visualize the object/s from different sides and applied dynamic view for parallel and perspective projection by using a camera and target with the help of DVIEW command. In this unit we also learnt the very interested command ANIPATH for motion path animation along the XY plane, and saved the animation file of a camera moving and panning in a 3D model.

- In the third unit of this module we learnt the constitution of 3D wire-frame, surfaces and 3D solids which were already discussed in previous module and will be discussed in more detail in later module.

Frequently Asked Questions (FAQs)

FAQ 1:	How could you display Animation Path in render Tab if Animation panel is not displayed?
Answer	If the Animations panel is not displayed on the Visualize tab, right-click the Visualize tab and click Panels →Animations.
FAQ 2:	How could you display Camera in render Tab, if camera option is not displayed?
Answer	If the camera option is not displayed on the Visualize tab, right-click the Visualize tab and click Panels→camera.
FAQ 3:	Can you edit objects while the 3DFORBIT command is active?
Answer	Objects cannot be edit when 3DFORBITcommand is active.
FAQ 4:	Can you construct surfaces and 3D solids from 2d geometry?
Answer	Yes! I can construct surfaces and 3D solids from 2D geometry by extruding, sweeping, lofting, and revolving.
FAQ 5:	How to Switch between parallel and perspective when 3dorbit command is active?
Answer	Switching between parallel and perspective view can be done when 3dorbit command is active by right click in the drawing and select parallel or perspective options.
FAQ 6:	Is there any more formats other than WMV to save motion path animation file?
Answer	Yes! Other formats are AVI and MPG
FAQ 7:	Is there any option to reverse motion Path Animations?
Answer	Yes! It can reverse motion path animation by check the reverse check box in the motion Path Animations Dialog box.
FAQ 8:	Can Animated saved file be open and move to other location?
Answer	Yes! Animated saved file can be easily open without Opening AutoCAD application and can be easily move and copy to other location.

Test Yourself!

Please mark the correct one from the given options. You can check your answer with the Answer Key at the end of this module.

1. 3D Orbit is used to revolve?

a. Entire Drawing
b. Only selected object
c. Selected and entire drawing
d. None

2. Short cut key for Free Orbit_____.

a. Ctrl + Click
b. 3DFORBIT
c. FORBIT
d. Both b and c

3. Fly orbit works in_____.

a. Parallel view
b. Only Perspective view
c. Both Parallel and perspective
d. None

4. ECS button is used to?

a. Reset the view
b. Delete the object
c. Exit the active command
d. None

5. Command for zoom orbit?

a. z
b. 3dz
c. 3dzoom
d. Both b and c

Answer Key

MCQ Number	Correct Answer
1	c
2	b
3	b
4	c
5	c

AUTOCAD

Module-6

LEARNING GUIDE

National Vocational Certificate Level 3

Version 1 - July 2013

Module 6: Produce 2D Solids and 3D Faces

Learning Outcomes

After completion of this learning module, you will be able to:

- Draw 2D Solids and 3D Faces
- Draw Edges

Learning Unit 1: Draw 2D Solids and 3D Faces

Overview

The unit explores the two dimensional solids and three dimensional faces, after completing this unit you will be able to execute “SOLID” Command with points to fill triangles and quadrilaterals and more shapes and objects.

After completion of this learning unit, you will be able to execute “3DFACE” command with points to fill objects. 3DFACE creates a three or four sided surface anywhere in 3D space differs from SOLID, which creates a three- or four-sided surface that is parallel to the current UCS and can be extruded.

Further you will also be able to create edges with 3DFACE, and creates a three-dimensional polyface mesh vertex by vertex.

1. Introduction OF 2D Solids

Solid Command is used to create 2D filled polygons with a color. These polygons can be solid-filled triangles and quadrilaterals we will discuss in this unit.

Remember

Polygonal:

A closed plane figure bounded by three or more line

Procedure

Create 2D Solid

STEP BY STEP Operations on AutoCAD

GET READY: Before you begin these steps, be sure to set Units as per your nature of work.

Note: Solid command is looked like Hatch but easier than the hatch.

Procedure

To Fill 2D Triangle by Solid Command

Step-1: Type **SOLID** Command in command bar <Enter>.

Follow Solid command Instructions

Step-2: First point: **P1**

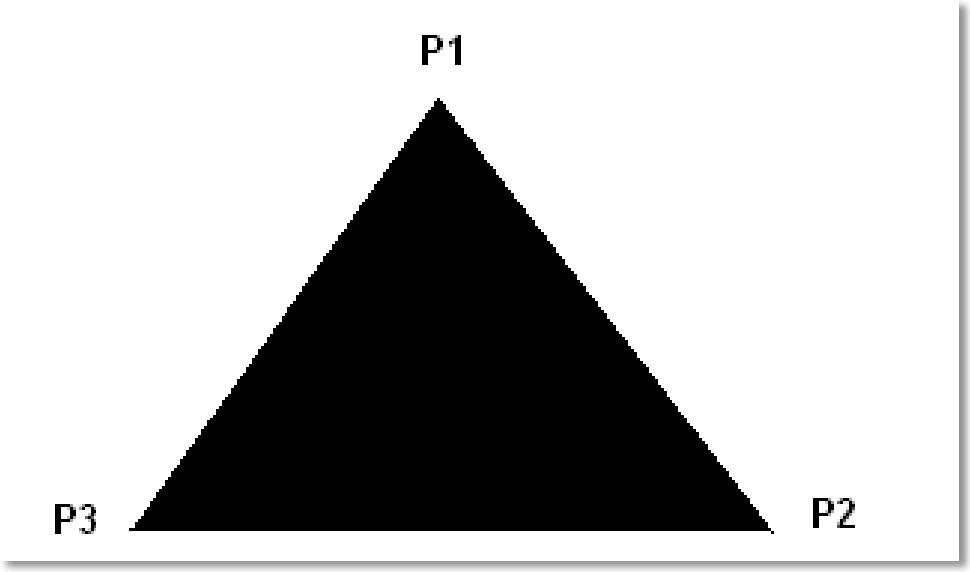
Step-3: Second point **P2**

Step-4: Third point: **P3**

Step-5: Fourth point: **<Exit>**Click the Right Mouse Button to finish solid.

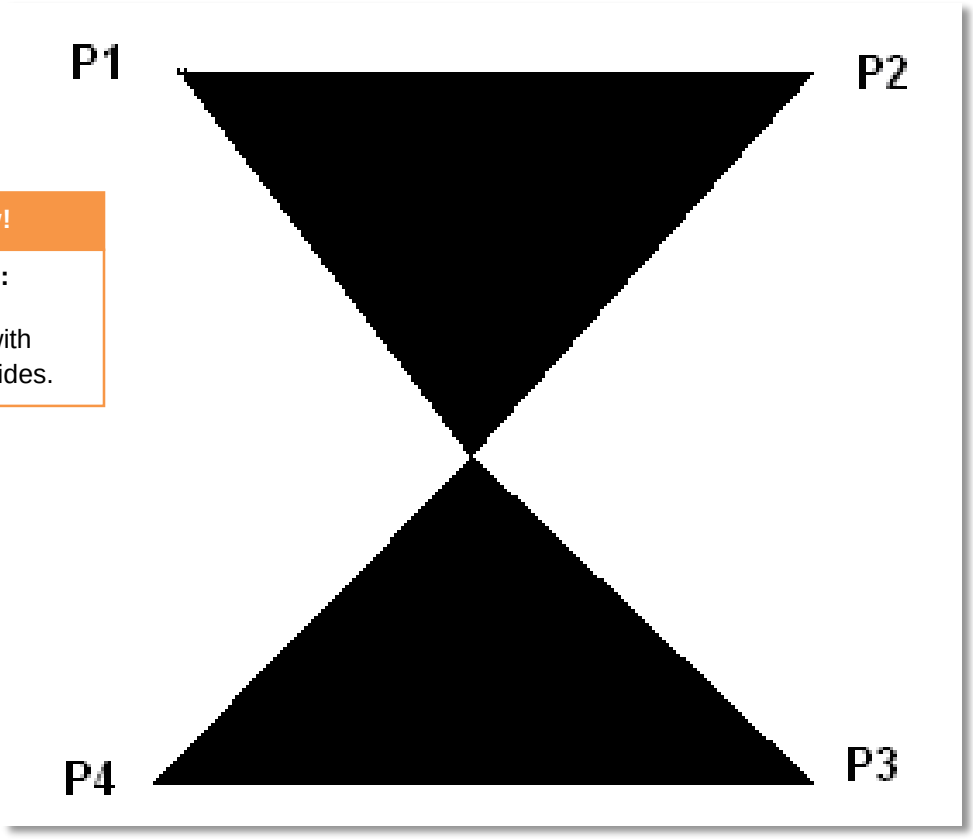
Do you know!

Polygon:
A plane figure with at least three straight sides and angles, and typically five or more.



Explanation:

If you follow the Instructions till fourth point then you will get the quadrilateral shape as follow below.



Do you know!

Quadrilateral:

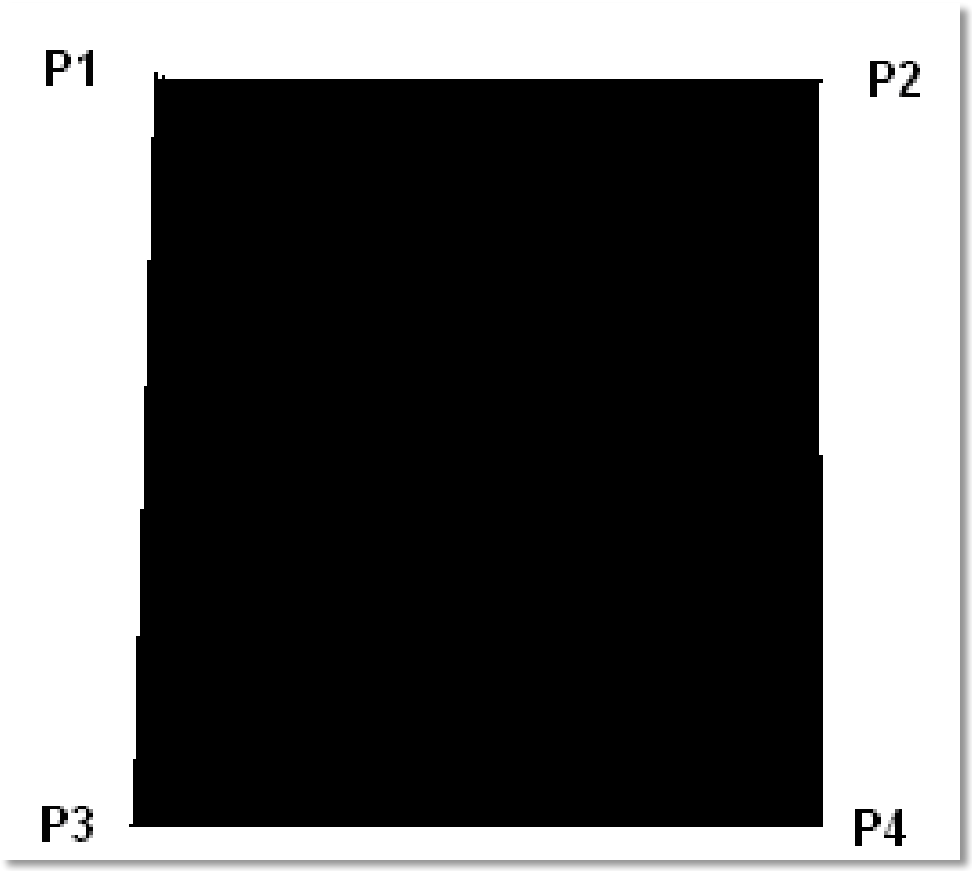
A flat shape with four straight sides.

When you create a quadrilateral solid-filled area, the sequence of the third and fourth points determines its shape.

The above 2D Solid shape tells us the pattern; the first two points define one edge of the polygon. The third point is diagonally opposite the First point and the fourth point is diagonally opposite the Second point

While:

If you follow the instructions till fourth point with different pattern, then you will get the rectangle shape as shown in the figure below.



The above 2D Solid shape tells us the pattern; the first two points define one edge of the polygon. The third point is diagonally opposite the second point and the fourth point is diagonally opposite the first point.

Fill Mode: 2D SOLIDS fill can be on or off by using **fill** command in AutoCAD.

2D solids are filled only when the **FILLMODE** system variable is **ON(1)** otherwise you will not fill any polygonal shapes by using solid command.

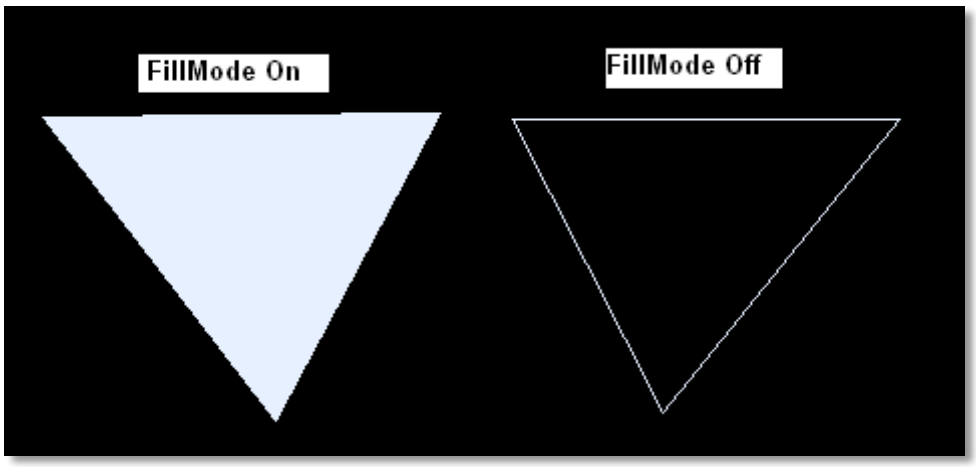
Procedure

FILLMODE ON/OFF

Step-1: Type **fill/FILL** command in command bar <EnterStep-2: Type OFF (by default it set to ON (1))

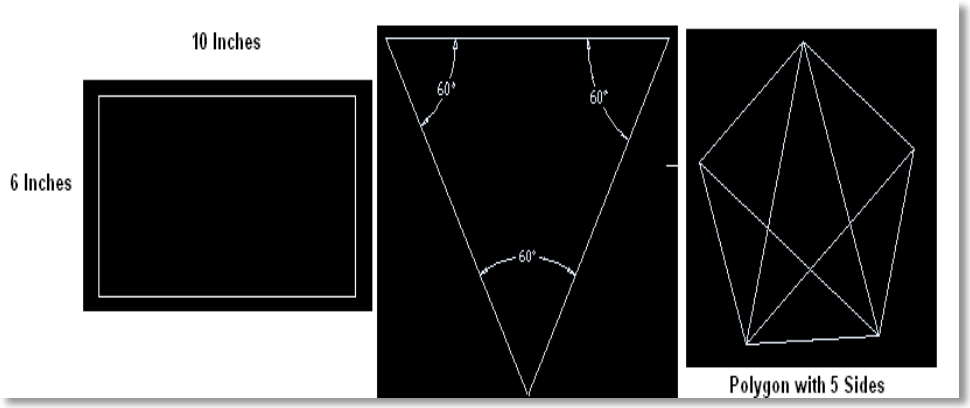
Now you will not fill area by using Solid command.

The Figure below show both **FillMode ON** and **FillMode OFF**



Learner's Activity

Apply Solid command to fill the given two shapes:



Solid-Filled Areas

There are several methods for creating solid-filled areas.

Solid-filled areas can be created using:

- Hatches with a solid hatch pattern (HATCH)
- 2D solids (SOLID)
- donuts (DONUT)

2. 3D Faces

Definition: 3D Face creates a three-sided or four-sided surface in 3D space.

Or

3D Face Command is used to draw 3D Surfaces with 3 or 4 edges in complete

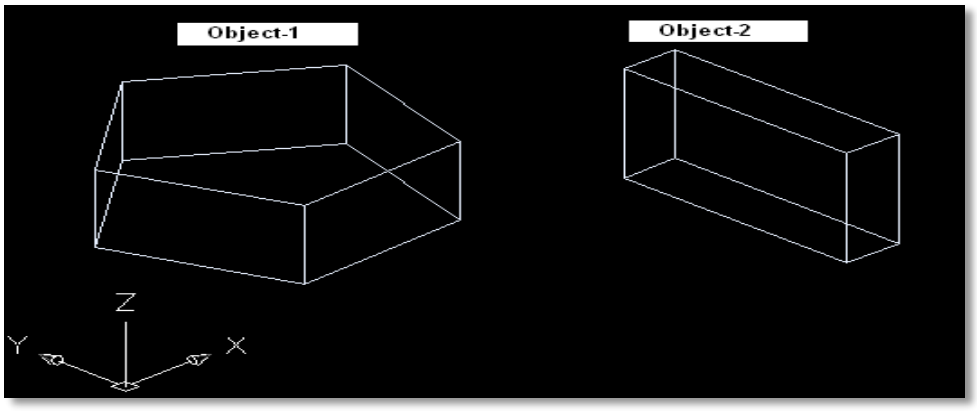
3D environment with **Z** coordinates.3D face creates a planar mesh with either three or four sides.

Command Entry
3Dface

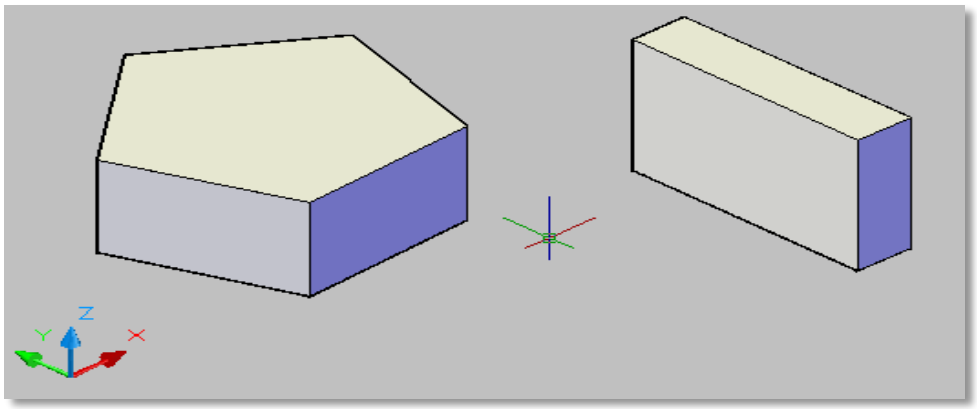
Procedure

Apply 3D Faces

We have two objects here to create 3D Surface by using 3D Face command on it.



Step-1: For Smooth Operation, you need to change Visual Style into Conceptual, as shown in the figure below

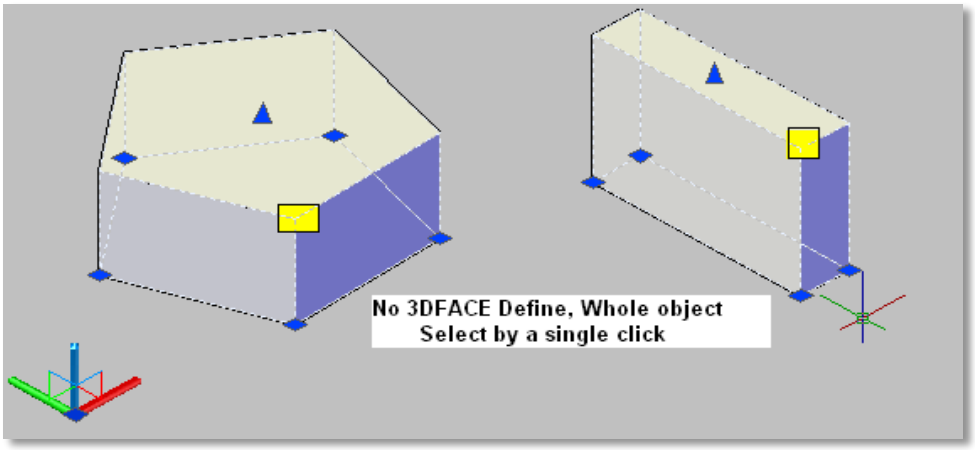


Step-2: Before Applying 3DFACE command, you need to click the objects at any point; you will get the object to be selected as a whole, as shown in the figure below.

Do you know!

Invisible:

You can Control the edges of 3D face by using I or invisible for the point to make Invisible that point while you want to get the 3D Surface.



Step-3: Type **3Dface** Command in command bar <Enter>

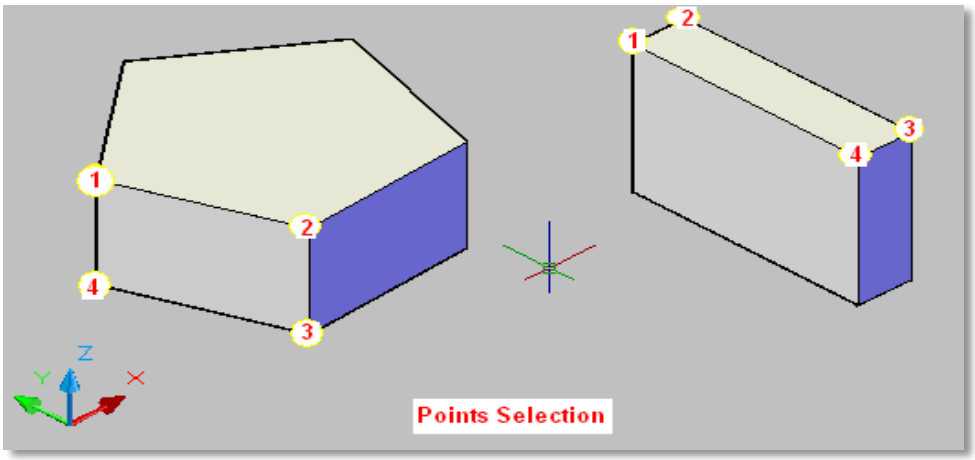
Step-4: Specify first point or [Invisible]:

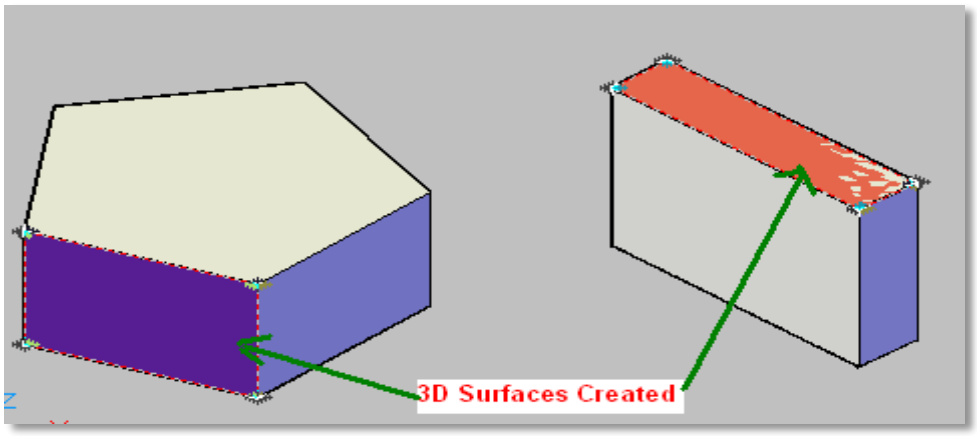
Step-5: Specify second point or [Invisible]:

Step-6: Specify third point or [Invisible] <exit>:

Step-7: Specify fourth point or [Invisible] <create three-sided face>:

Step-8: Specify third point or [Invisible] <exit>: *Cancel*





Step-9: You can select the 3D Surface individually, and can apply different shade, color and render to it as shown in the figure above.

Learner's Activity

Create a 3D Surface for the top surface of an **object-1** by using 3DFACE, but the Point 2 and point 5 must be Invisible.

3. PFaces

Another technique to apply 3d face by using **PolyFace**, it is used to creates a three-dimensional polyface mesh vertex by vertex.

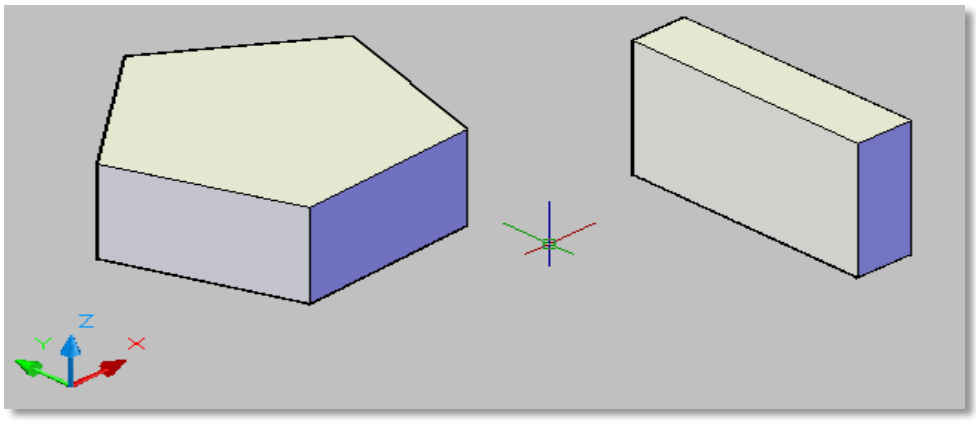
Color can be applied while Faces created with PFACE adopt the current layer and color. Unlike polyline vertices, polyface mesh faces can be created with layer and color properties different from their parent object.

Command Entry
Pface

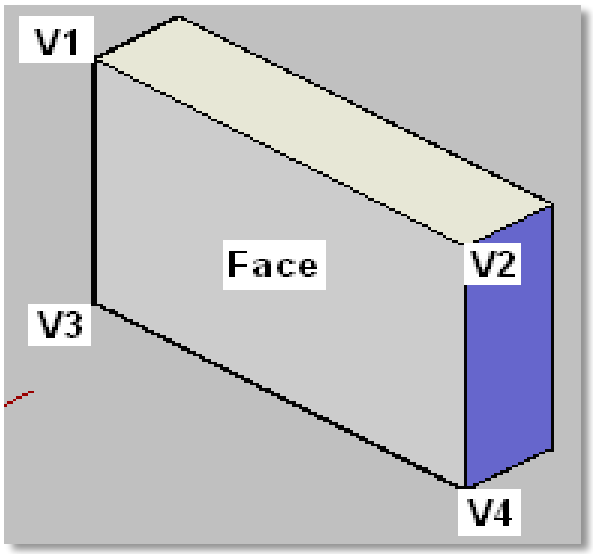
Procedure

Apply Poly Faces

Apply Polyface command to the following object to create a 3D polyface mesh vertex by vertex.



Polyface with one face and 4 Vertex.



Step-1: Type **PFACE** in command bar <Enter>

Step-2: Specify location for vertex 1: **V1**

Step-3: Specify location for vertex 2 or <define faces>: **V2**

Step-4: Specify location for vertex 3 or <define faces>: **V3**

Step-5: Specify location for vertex 4 or <define faces>: **V4**

Step-6: Specify location for vertex 5 or <define faces>: **Enter**

Step-7: Face 1, vertex 1:

Step-8: Enter a vertex number or [Color/Layer]: 1

Face 1, vertex 2:

Step-9: Enter a vertex number or [Color/Layer] <next face>: 2

Face 1, vertex 3:

Step-10: Enter a vertex number or [Color/Layer] <next face>: 3

Face 1, vertex 4:

Step-11: Enter a vertex number or [Color/Layer] <next face>: 4

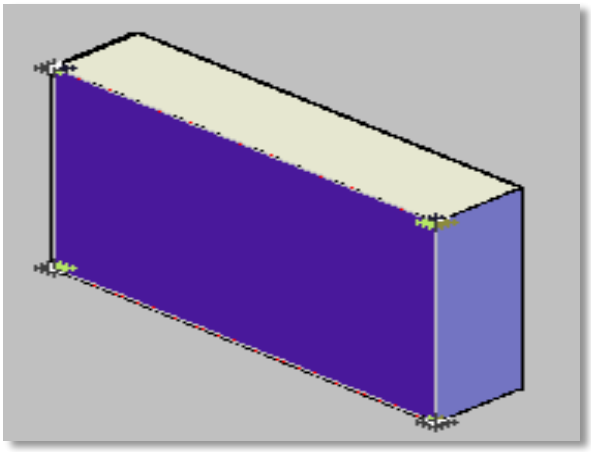
Face 1, vertex 5:

Step-12: Enter a vertex number or [Color/Layer] <next face>: **Enter**

Step-13: Face 2, vertex 1: **Enter <There is only one face, so Need to Press Enter>**

Step-14: Enter a vertex number or [Color/Layer]: <If you want to Apply color then type **c** or **L**> **Enter**

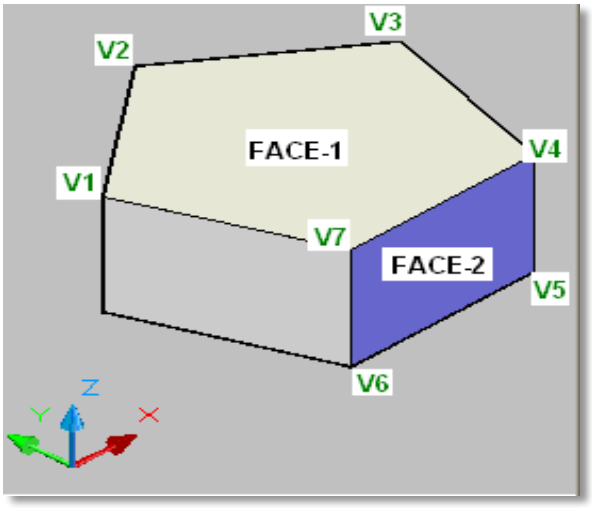
Output as shown in the figure below:



Learner's Activity

Create Poly faces for the following object.

Polyface with two faces and 7 Vertex.



Hints:

Face-1: V1, V2, V3, V4 and V7

Face-2: V4, V5, V6 and V7

Learning 2: Unit Edge

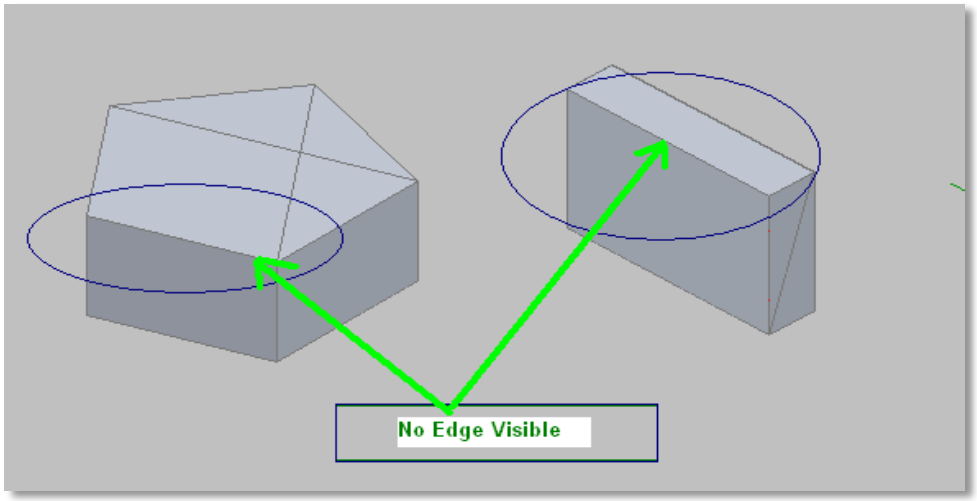
Overview

After completing this unit, you will be able to execute Edge command with toggle visibility and hidden edges and also be able to draw 3D faces with invisible edges and make Invisible edges visible or display by using the sub command of Edge, edge and display.

1. Edge

EDGE command is used to control the visibility. It will be created 3D FACE command is used to high light 3D face.

Command Entry
edge



Procedure

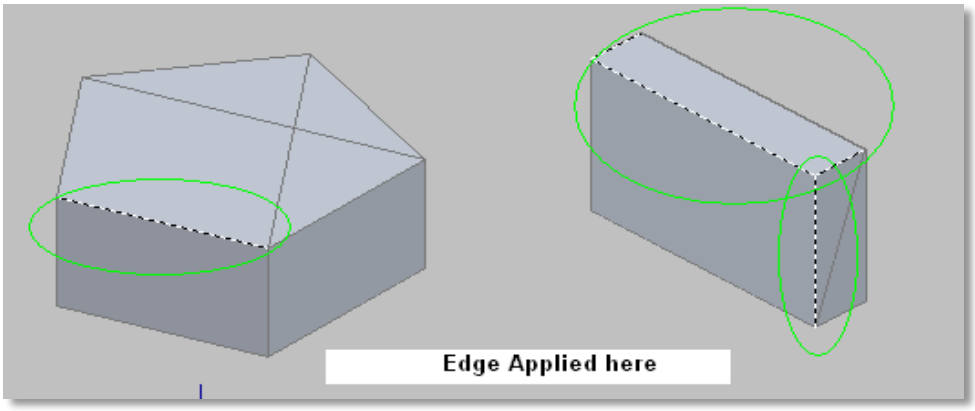
Apply Edge

Step-1: Type edge command in command bar <Enter>

Step-2: Specify edge of 3dface to toggle visibility or [Display]: **d**

Step-3: Enter selection method for display of hidden edges [Select/All] <All>: **a**

Regenerating 3DFACE objects...done.



Output after Edge applied

Learner's Activity

The edge command applied with display option, you need to apply edge selection method to display edges of the 3D Faces if created.

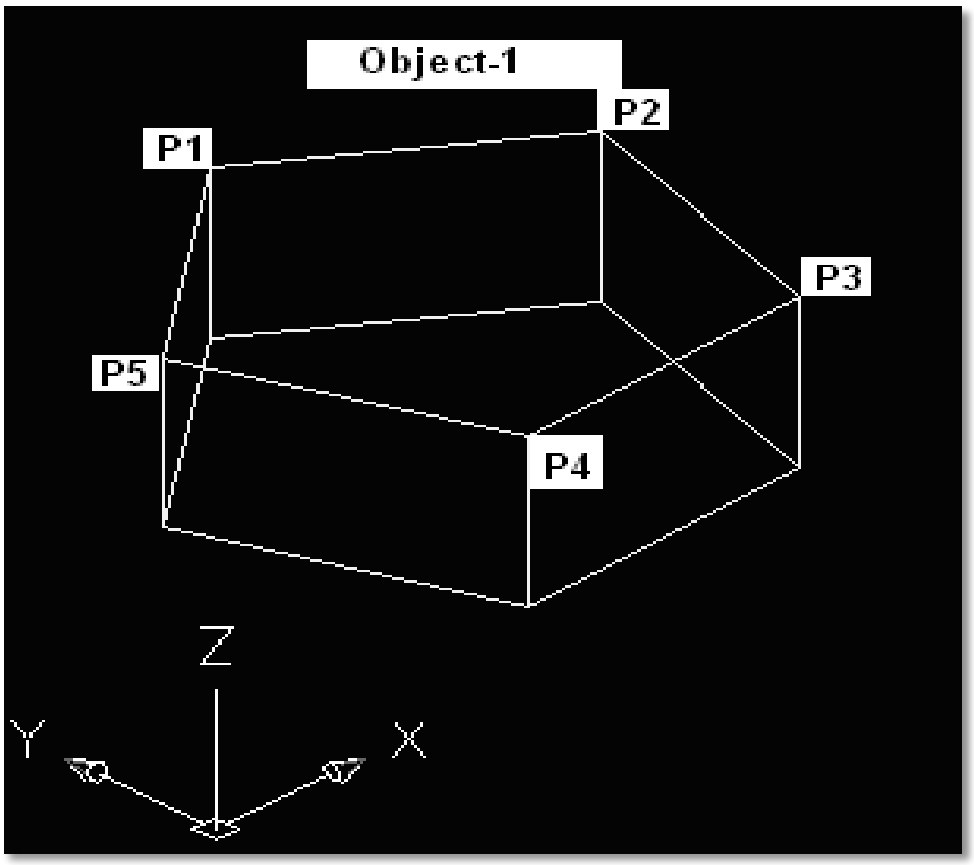
2. Draw 3D Faces with Invisible Edge

Invisible edges means to make 3D faces with not showing that faces.

Procedure

Apply Invisible Edge

As we draw 3D Faces in unit-1 of this module with visible edges, now we are going to draw again those edges but with the option of "i" or Invisible command to makes the edge invisible, further we will apply edge command to makes the edge visible or display.



Step-1: Change the Visual style view into realistic view

Step-2: Enter **3dface** command in command bar <Enter>

Step-3: Specify first point or [Invisible]: **P1**

Step-4: Specify second point or [Invisible]: **P2**

Step-5: Specify third point or [Invisible] <exit>: **I**

Step-6: Specify third point or [Invisible] <exit>: **P3**

Step-7: Specify fourth point or [Invisible] <create three-sided face>: **I**

Step-8: Specify fourth point or [Invisible] <create three-sided face>: **P4**

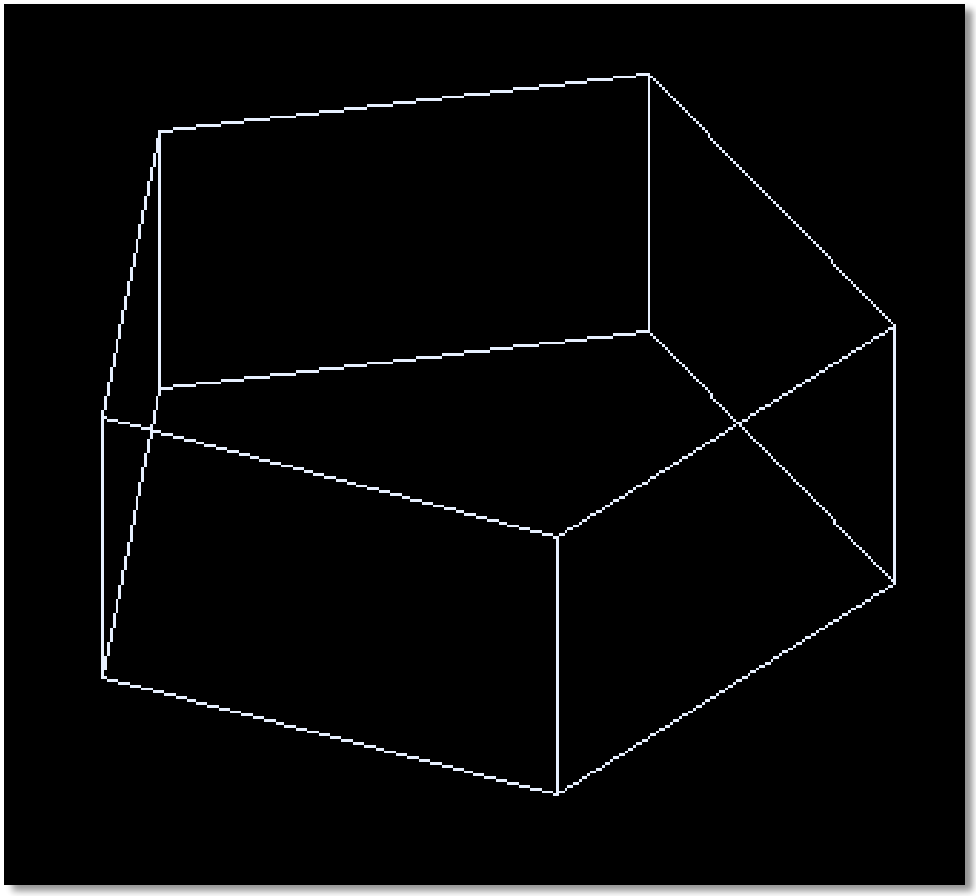
Step-9: Specify third point or [Invisible] <exit>: **I**

Step-10: Specify third point or [Invisible] <exit>: **P5**

Step-11: Specify fourth point or [Invisible] <create three-sided face> : **<Enter>**

Step-12: Specify third point or [Invisible] <exit> : **<Enter>**

The Output should be look like



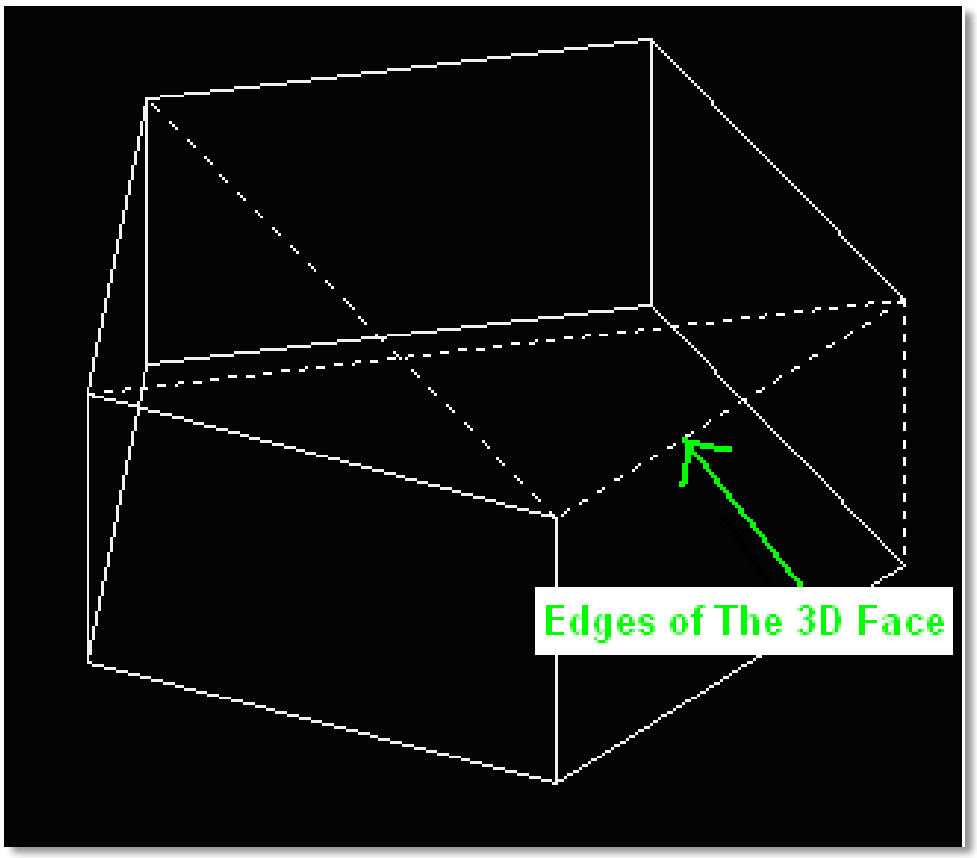
Now to make the edge Visible or display.

Step-1: Type edge command in command bar <Enter>

Step-2: Specify edge of 3dface to toggle visibility or [Display]: **d**

Step-3: Enter selection method for display of hidden edges [Select/All] <All>: **a**

Output after Edge applied


Edges of The 3D Face

Summary of Module

- In this module we learnt the concept of solid-filled area by using its pre-defined command in AutoCAD, filled two types of polygonal areas like triangular areas and quadrilateral area with different colors and FILLMODE ON/OFF concepts as well. Further we successfully covered making of 3D Faces with Invisible and visible edge concepts by using pre-defined command **3Dface** and **edge**.

- We also apply another technique to apply 3DFACE by using **Pface** command, and created a three-dimensional polyface mesh vertex by vertex.

Frequently Asked Questions (FAQs)

FAQ 1:	Is there any command to fill area other than solid command?
Answer	Yes! Hatch command is used to fill an enclosed area.
FAQ 2:	What is the Purpose of Hatch command?
Answer	Purpose of hatch command is to fill an enclosed area or selected objects with a hatch pattern, solid fill, or gradient fill.
FAQ 3:	What is the Purpose of DONUT command?
Answer	Purpose of DONUT command is to draw filled circles and rings.
FAQ 4:	How to makes edge invisible?
Answer	Entering “I” or invisible before the point of an edge makes the edge invisible.
FAQ 5:	What is the purpose of Select option in edge command?
Answer	Purpose of select option is to select the hidden edge and to make the edge visible partially.

Test Yourself!

Please mark the correct one from the given options. You can check your answer with the Answer Key at the end of this module.

1. Solid command never fills area when……

a. Fillmode is off
b. Area never enclosed
c. Object having less than two sides
d. Above all

2. To make quadrilateral solid-filled area, follow solid command till

a. 2nd point
b. Fourth point
c. Third point
d. Both b and c

3. To make triangular solid-filled area, follow solid command till

a. 2nd point
b. Forth Point
c. Third point
d. Both b and c

4. Command to ON/OFF Fill Mode system variable

a. Filled
b. FM
c. Fill
d. F

5. 3DFace works in

a. Conceptual view
b. Realistic view
c. 3d hidden view
d. All

Answer Key

MCQ Number	Correct Answer
1	d
2	b
3	c
4	c
5	a

AUTOCAD

Module-7

LEARNING GUIDE

National Vocational Certificate Level 3

Version 1 - July 2013

Module 7: Insert Surfaces

Learning Outcomes

After completion of this learning module, you will be able to:

- Draw Basic Surfaces
- Draw Complex Surfaces

Learning Unit 1: Draw Basic 3D Surfaces Object

Overview

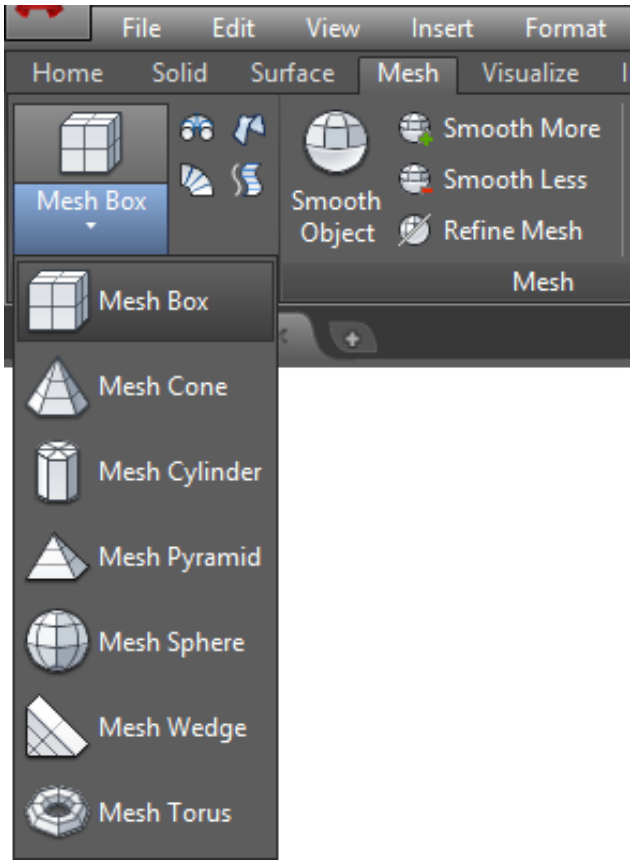
This unit describes about fundamental mesh primitive commands by which you can reshape mesh objects. At the end of this unit, you will be able to perform various Mesh commands such as mesh box use to create rectangular or cubical mesh box, Mesh cone use to draw cone, Mesh cylinder use to create cylinder of varying radius, Height, Mesh pyramid of inscribed / circumscribed perimeter, Mesh wedge and Mesh torus which use to create ring shape surface of two radius value. You will be able to adjust the smoothness of mesh by applying more or less smooth mesh command. You can also able to add or remove mesh crease which helps to sharpen or crease the edges of mesh objects. Editing mesh commands such as Mesh extrude, Mesh split, Merge face, Mesh cap, and helps you to edit mesh as required.

1. Basic Mesh Commands

The basic mesh forms, known as mesh primitives, are the equivalent of the primitive forms for 3D solids. You can reshape mesh objects by smoothing, creasing, refining, and splitting faces. You can also drag edges, faces, and vertices to mold the overall form.

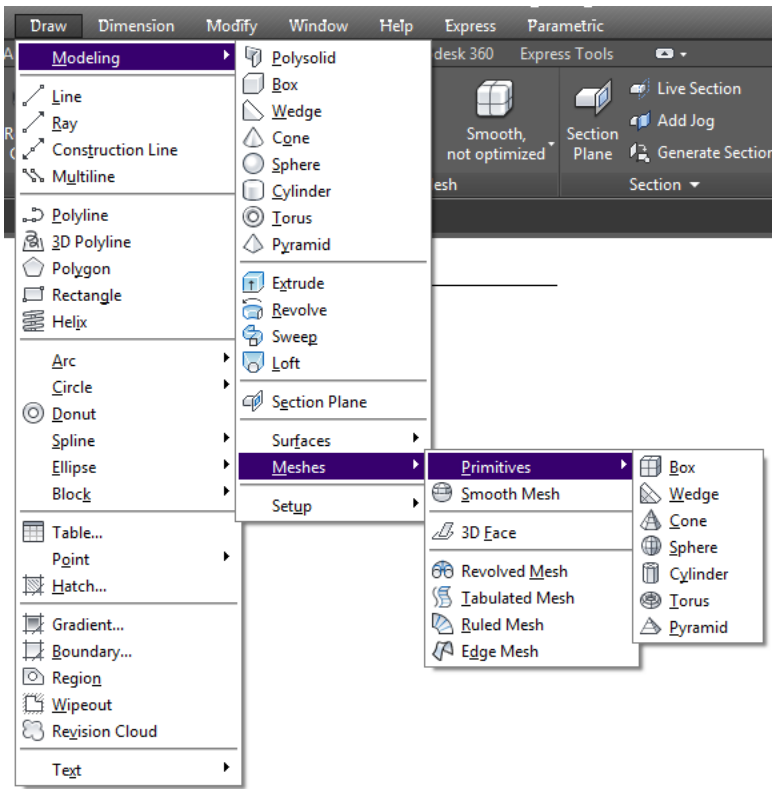
Basic Mesh commands can be find out through following ways:

Through Ribbon drop down Menu



Through Main Menu

Main Menu→Draw→Modeling →Meshes→Primitives →Show Mesh Tools



Show Mesh Primitive Options Dialog Box

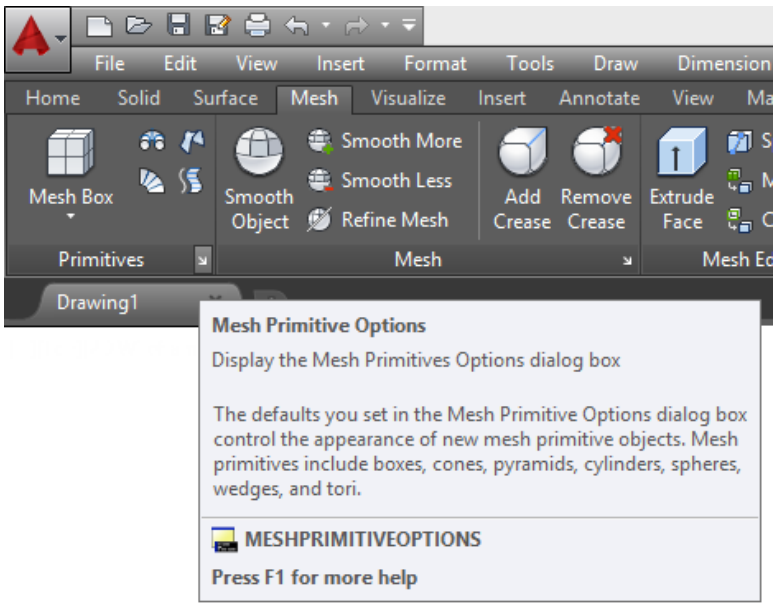
Click the Mesh tab→Click the Primitives panel

→Show Mesh Primitive Options Dialog Box

Remember

Tessellation Divisions

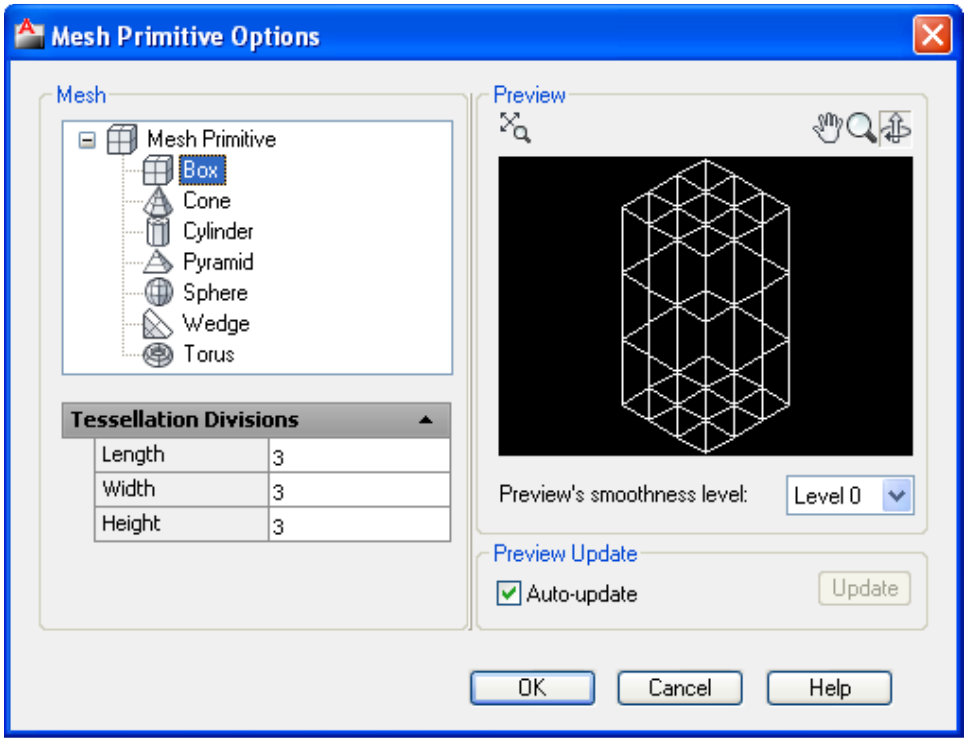
For each selected mesh primitive type, sets the default number of divisions for each side. Enter a new value to reset the number of divisions for new mesh primitives.



Through Mesh Primitive Options Dialog Box

The following settings can be adjusted for mesh primitives. The defaults you set in the Mesh Primitive Options dialog box control the appearance of new mesh primitive objects. Mesh primitives include boxes, cones, pyramids, cylinders, spheres, wedges, and torus.

This dialog box is also available from the Mesh and Tessellation Divisions of mesh primitive



2. Create Mesh Commands

1) **Mesh Box**
2) **Mesh Cone**
3) **Mesh Cylinder**
4) **Mesh Pyramid**
5) **Mesh Sphere**
6) **Mesh Wedge**
7) **Mesh Tours**

Remember

By default, new mesh primitives are created with no smoothness. To change the default smoothness, enter mesh at the Command prompt. Specify the Settings option before you specify the type of mesh primitive you want to create.

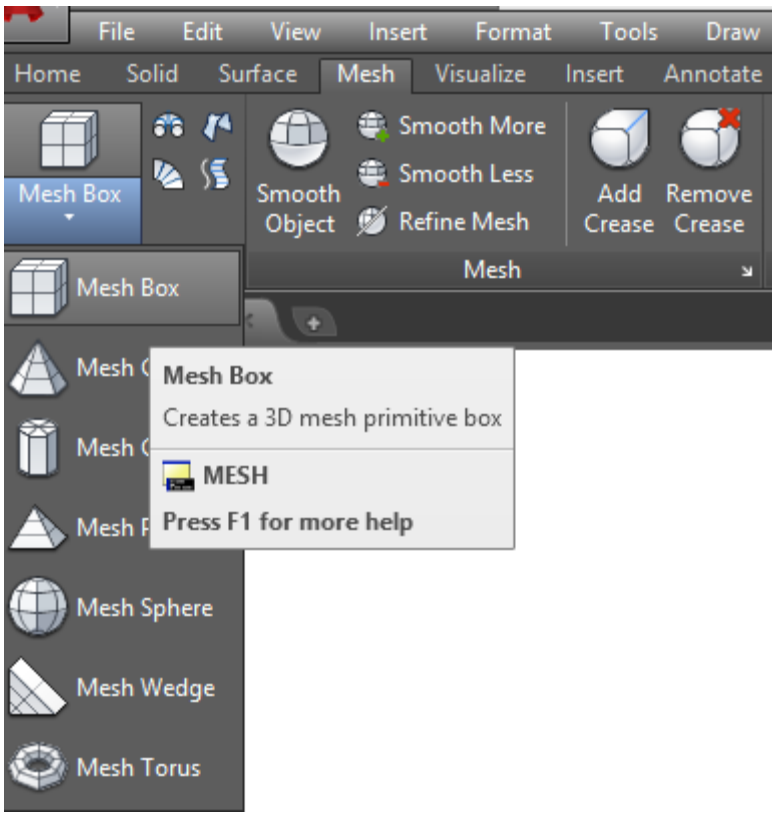
1) Mesh Box

Creates a rectangular or cubical mesh box. The base of the mesh box is drawn parallel to the XY plane of the current UCS (work plane).

Mesh Box command of 3D Surface can be find out through the following ways:

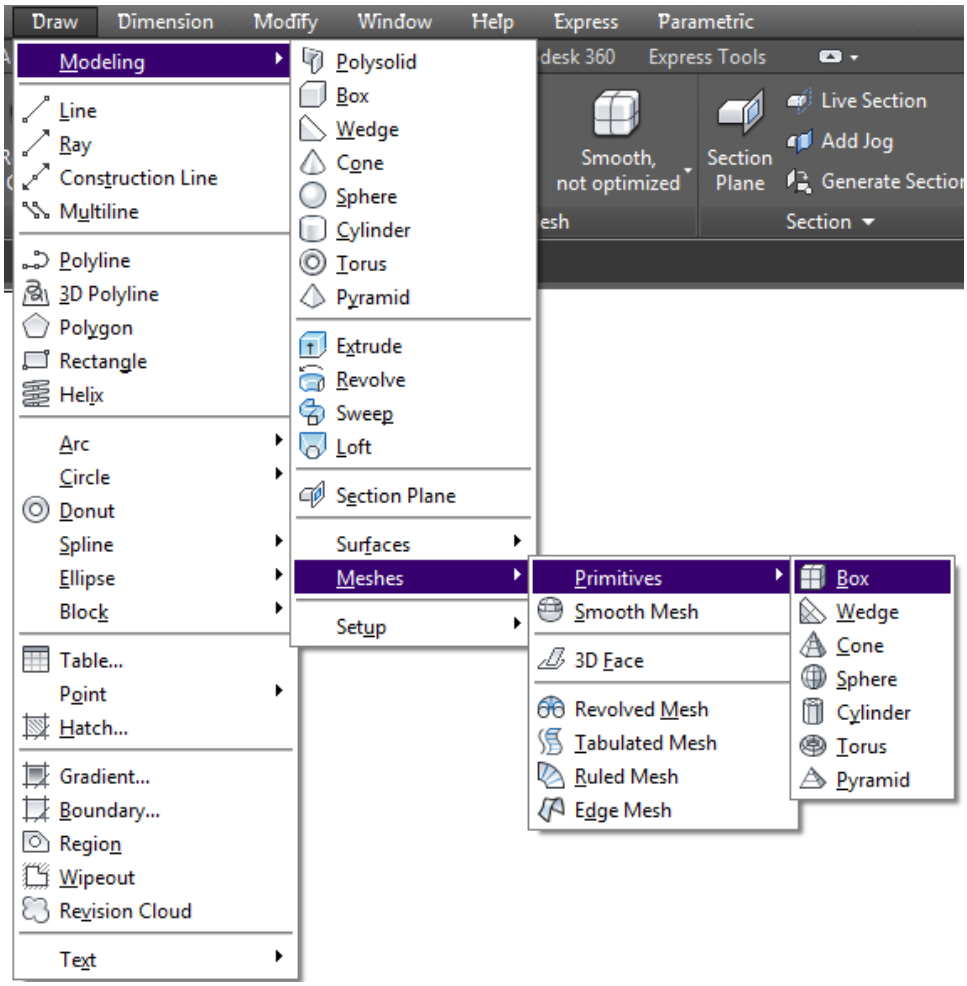
Through Ribbon

This figure describes the location of Mesh Box command of 3D Surface in Mesh Tab.



Through Main Menu

Main Menu →Draw →Modeling →Meshes→Primitives →Box



Through Short Cut Key

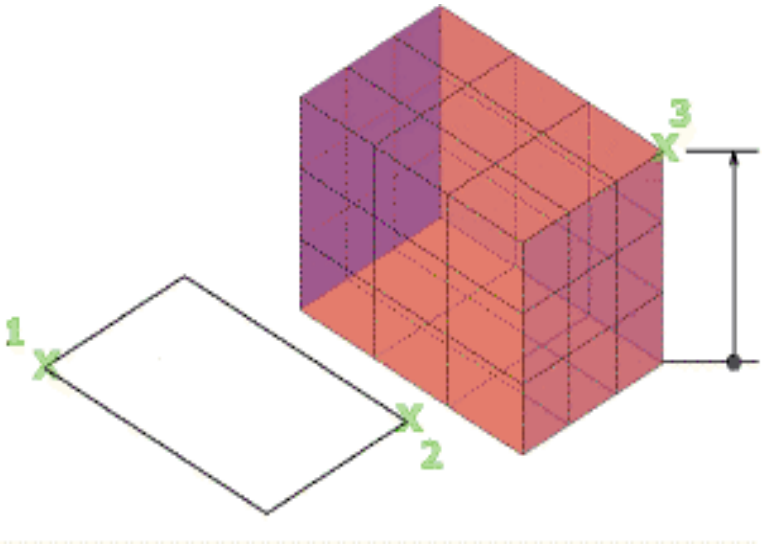
MESH enter → B enter

APPLICATION

Type MESH at the Command Prompt →[Box/Cone/Cylinder/Pyramid/Sphere/Wedge/Torus/SEttings] <Box>:→ B enter (for Box)→ Specify Start point→Put the Value of @ X and Y Co-ordinate→Put the Value of height →enter

Do you know!

You can use the Mesh Primitive Options dialog box to set the default number of divisions for a mesh box.



Mesh Box Options

The Box option of the MESH command provides several methods for determining the size and rotation of the mesh boxes you create.

i. Cube

ii. Length

iii. Center Point

i. Cube.

Use the Cube option to create a mesh box with sides of equal length.

ii. Length

Use the Cube or Length option if you want to set the rotation of the box in the XY plane.

iii. Center point.

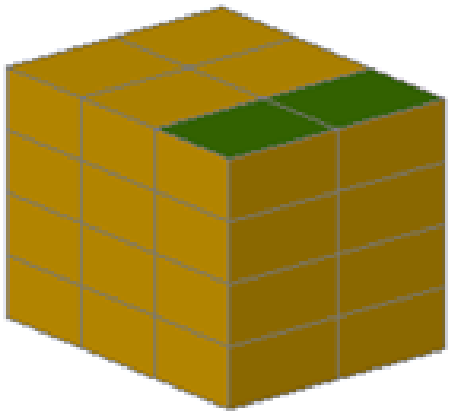
Use the Center option to create a box using a specified center point.

Tessellation Divisions of BOX

The following system variables are used to control the number of divisions for each dimension of a new mesh box:

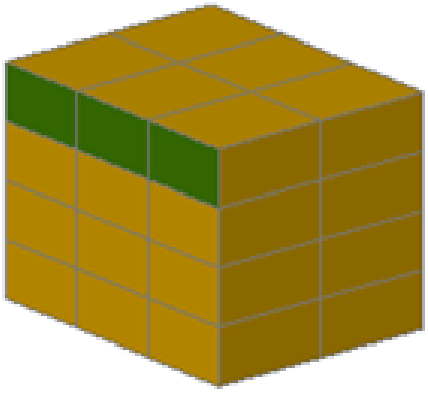
a) Length.

Set the number of divisions for the length of a mesh box along the X axis. (DIVMESHBOXLENGTH system variable)



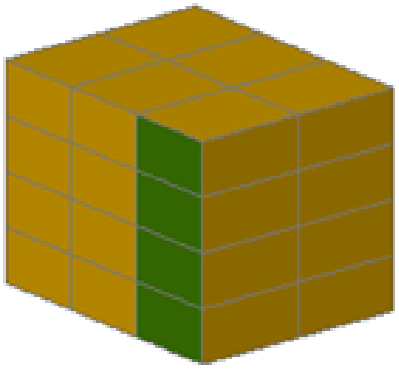
b) Width.

Set the number of divisions for the width of a mesh box along the Y axis. (DIVMESHBOXWIDTH system variable)



c) Height.

Set the number of divisions for the height of a mesh box along the Z axis. (DIVMESHBOXHEIGHT system variable)



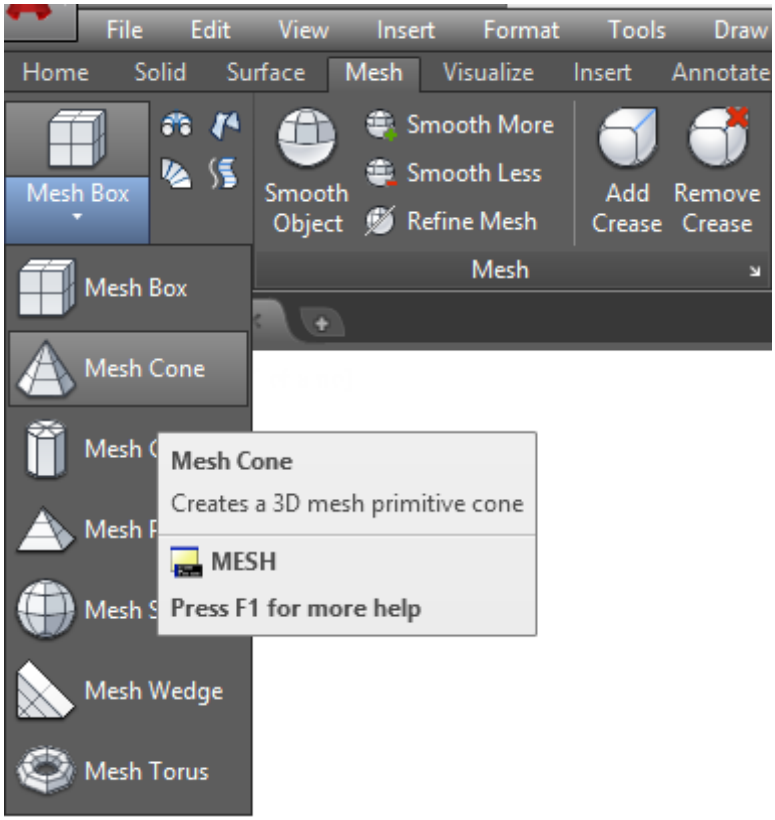
2) Mesh Cone

Creates a 3D surface with a circular or elliptical base that tapers symmetrically to a point or to a circular or elliptical planar face. You can control the smoothness of 3D curved surface, such as a cone, in a shaded or hidden visual style with the FACETRES system variable. Use the Top Radius option to create a cone frustum.

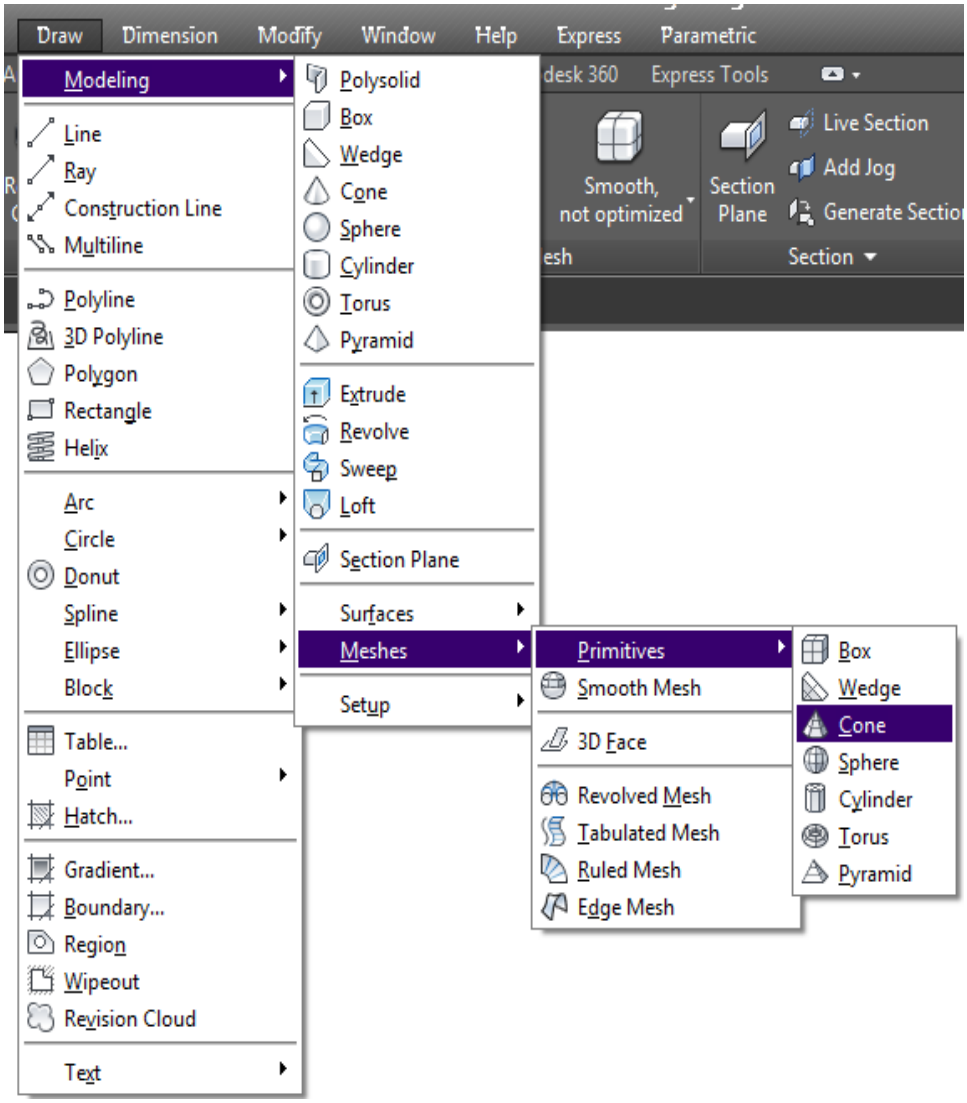
Mesh Cone command of 3D Surface can be find out through following ways:

Through Ribbon

This figure describes the location of Mesh Cone command of 3D Surface in Mesh Tab.



Through Main Menu
Main Menu →Draw →Modeling →Meshes→Primitives →Cone



Through Short Cut Key

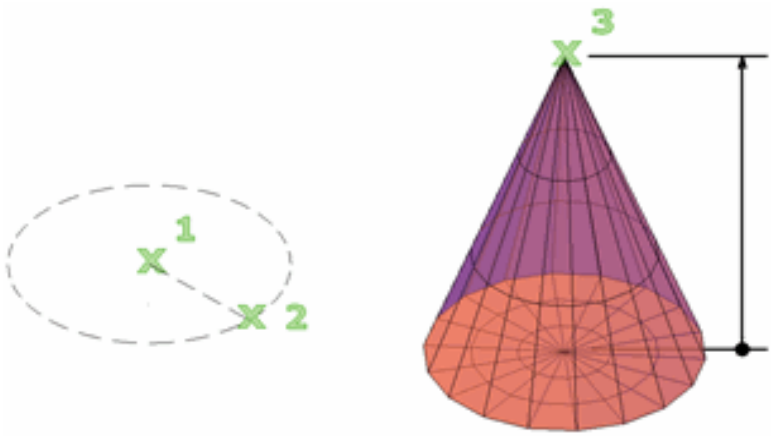
MESH enter → C enter

Application

Type MESH at the Command Prompt →[Box/Cone/Cylinder/Pyramid/Sphere/Wedge/Torus/SEttings] <Box>:→ C enter (for Cone)→ Specify center point of base→Put the base radius value→Put the Value of Height→enter

Remember

By default current smoothness level is always zero.



Mesh Cone Option

The Cone option of the MESH command provides several methods for determining the size and rotation of the mesh cones you create.

i. Center Point

ii. Axis Endpoint

iii. Frustum

iv. 3P

v. Elliptical

vi. TTR(Tangent, Tangent, Radius)

i. Center Point

Specifies the center point of the Cone. When you specify the center point, the Cone is positioned so that its central axis is parallel to the Z axis of the current user coordinate system (UCS). Latitudinal lines are parallel to the XY plane.

- **Radius** Defines the radius of the Cone.
- **Diameter** Defines the diameter of the Cone.

ii. Axis Endpoint

Use the Axis Endpoint option when you want to reorient the cone by placing the tip or axis endpoint anywhere in 3D space.

iii. Frustum

Use the Top Radius option to create a frustum of a cone, which tapers to an elliptical or planar face.

iv. 3P (Three Points)

The 3P (Three Points) option defines the size and plane of the base of the cone anywhere in 3D space.

v. Elliptical

Use the Elliptical option to create a cone base whose axes are of different lengths.

vi. TTR(Tangent, Tangent, Radius)

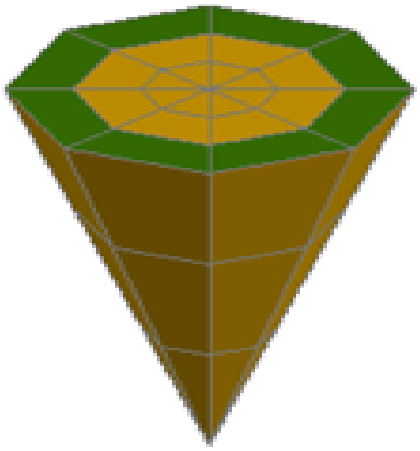
Use the TTR (Tangent, Tangent, and Radius) option to define points on two objects. Depending on the radius distance, the new cone is located as near as possible to the tangent points you specify. You can set up tangency with circles, arcs, lines, and some 3D objects. The tangency points are projected into the current UCS. The appearance of tangency is affected by the current level of smoothness.

Tessellation Divisions of Cone

The following system variables are used to control the number of divisions for each dimension of a new mesh cone:

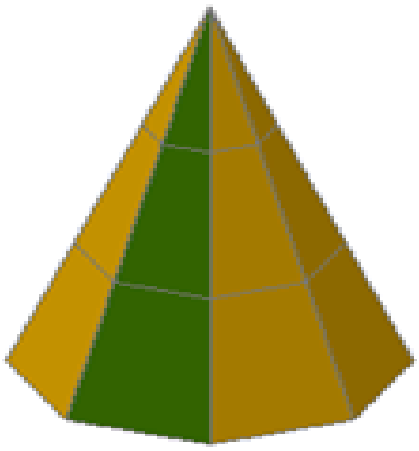
a) Axis.

Set the number of divisions around the perimeter of the mesh cone base. (DIVMESHCONEAXIS system variable)



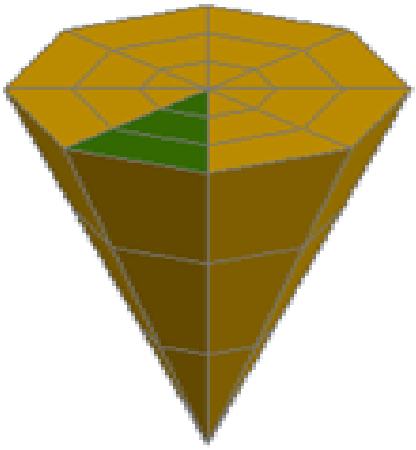
b) Height.

Set the number of divisions between the base and the point or top of the mesh cone.(DIVMESHCONEHEIGHT system variable)



c) Base.

Set the number of divisions between the perimeter and the center point of the mesh cone base.(DIVMESHCONEBASE system variable)



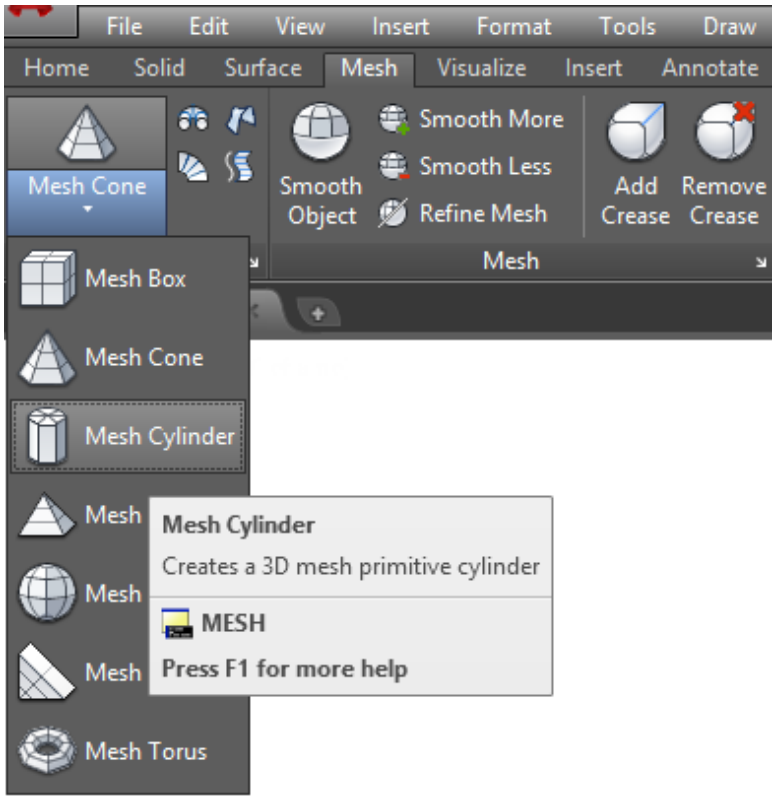
3) Mesh Cylinder

Creates a 3D surface with a circular or elliptical base that extends to the other circular or elliptical planar face. You can control the smoothness of curved 3D solids, such as a cylinder, in a shaded or hidden visual style with the FACETRES system variable.

Mesh Cylinder command of 3D Surface can be find out through following ways

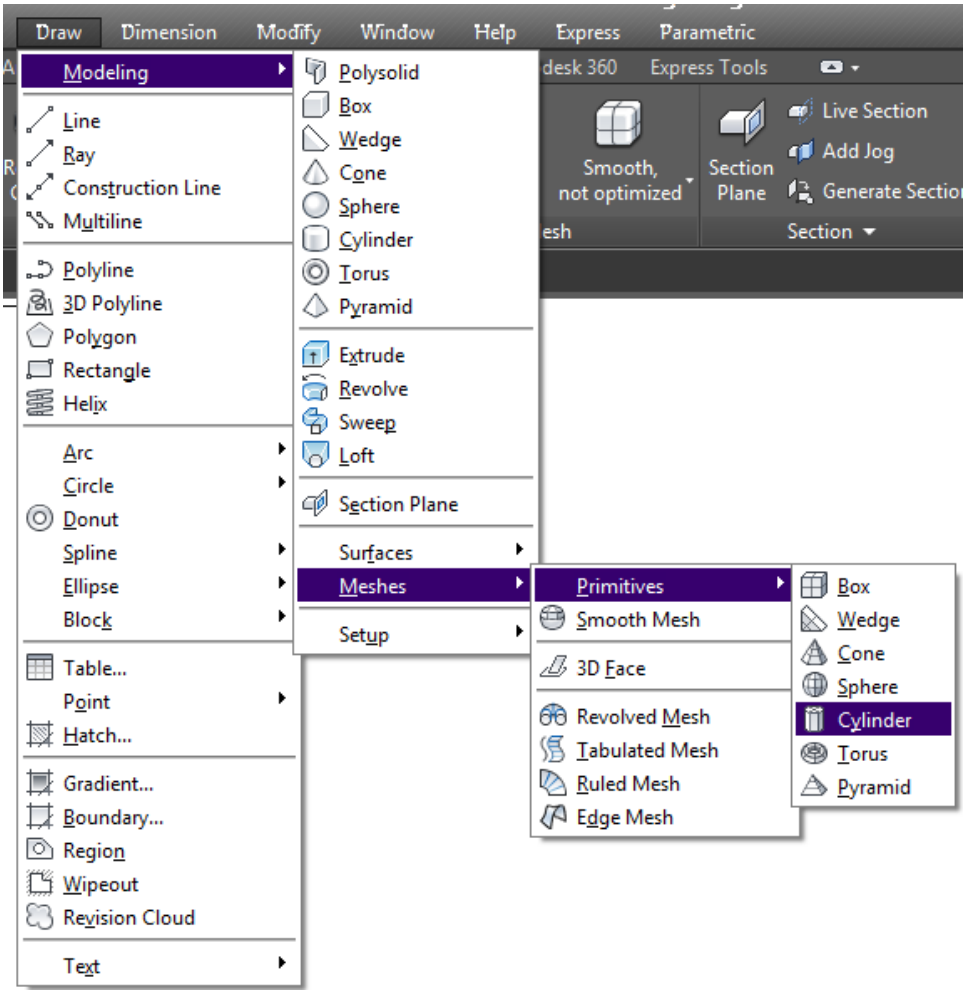
Through Ribbon

This figure describes the location of Mesh Cylinder command of 3D Surface in Mesh Tab.



Through Main Menu

Main Menu →Draw →Modeling →Meshes→Primitives →Cylinder

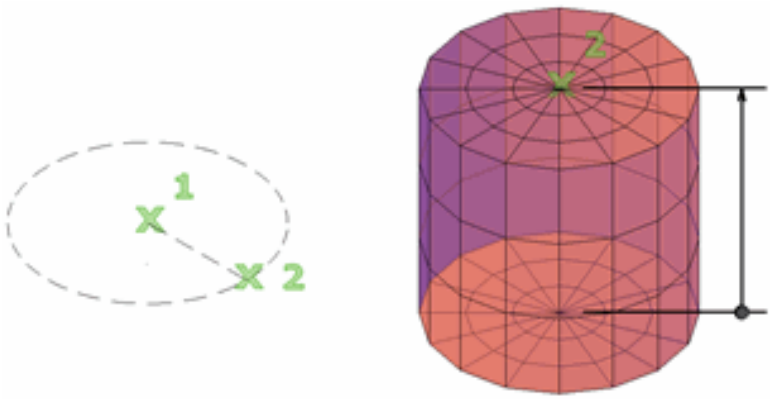


Through Short cut Key

MESH enter → CY enter

APPLICATION

Type MESH at the Command Prompt →[Box/Cone/CYlinder/Pyramid/Sphere/Wedge/Torus/SEttings] <Box>:→ CY enter (for Cylinder)→ Specify center point of base→Put the base radius value→Put the Value of Height →enter



Remember

During a drawing session, the default value for the base radius is always as the previously entered base radius.

Mesh Cylinder Options

The Cylinder option of the MESH command provides several methods for determining the size and rotation of the mesh cylinders you create

i. Axis Endpoint

ii. 3P(Three Point

iii. Elliptical

iv. TTR(Tangent, Tangent, Radius)

i. Axis Endpoint

Use the Axis Endpoint option to set the height and rotation of the cylinder. The center point of the top plane of the cylinder is the axis endpoint, which can be located anywhere in 3D space.

ii. 3P (Three Points)

Use the 3P (Three Points) option to define the base of the cylinder. You can set three points anywhere in 3D space.

iii. Elliptical

Use the Elliptical option to create a cylinder base whose axes are of different lengths.

iv. TTR(Tangent, Tangent, Radius)

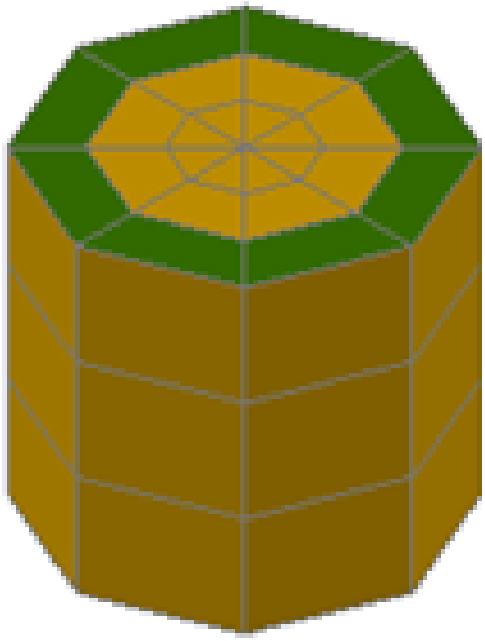
Use the TTR (Tangent, Tangent, and Radius) option to define points on two objects. Depending on the radius distance, the new cylinder is located as near as possible to the tangent points you specify. You can set up tangency with circles, arcs, lines, and some 3D objects. The tangency points are projected onto the current UCS. The appearance of tangency is affected by the current level of smoothness.

Tessellation Divisions of Cylinder

The following system variables are used to control the number of divisions for each dimension of a new mesh cylinder:

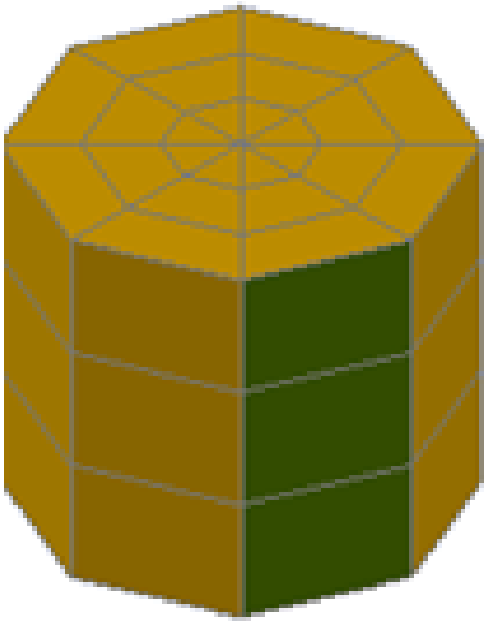
a) Axis.

Set the number of divisions around the perimeter of the mesh cylinder base.(DIVMESHCYLAXIS system variable)



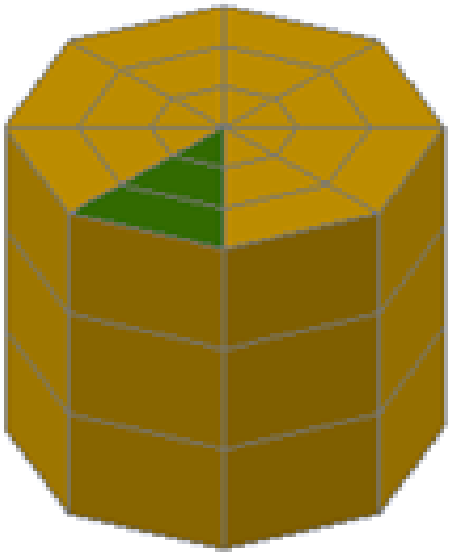
b) Height.

Set the number of divisions between the base and the top of the mesh cylinder.(DIVMESHCYLHEIGHT system variable)



c) Base.

Set the number of divisions between the perimeter and the center point of the mesh cylinder base. (DIVMESHCYLBASE system variable)



4) Mesh Pyramid

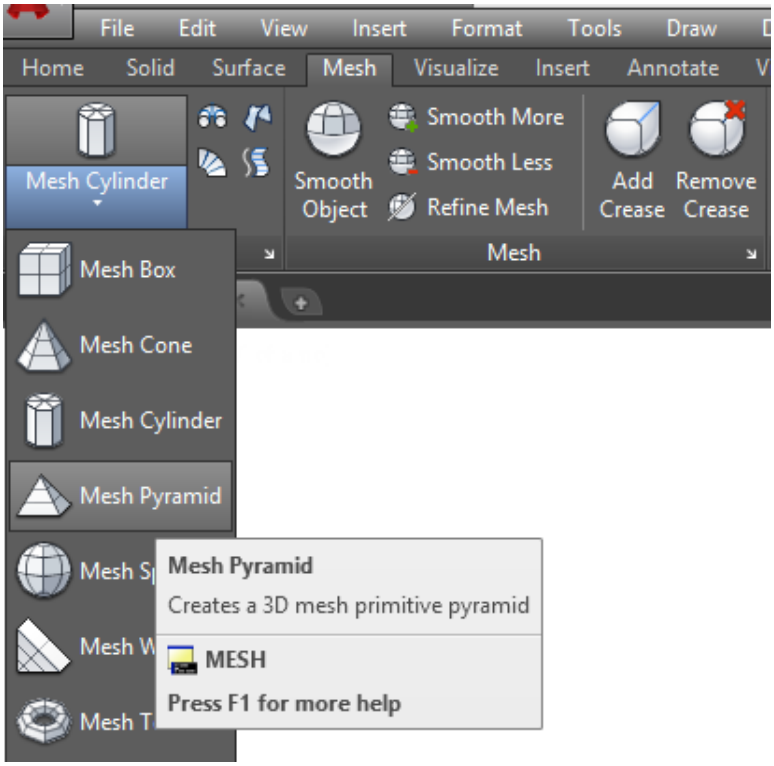
By default, a pyramid is defined by the center of the base point, a point on the middle of the edge, and another point that determines the height. Initially, the default base radius is not set to any value. During a drawing session, the default value for the base radius is always the previously entered base radius value for any solid primitive.

Use the Top Radius option to create a pyramid frustum.

Mesh Pyramid command of 3D Surface can be find out through following ways:

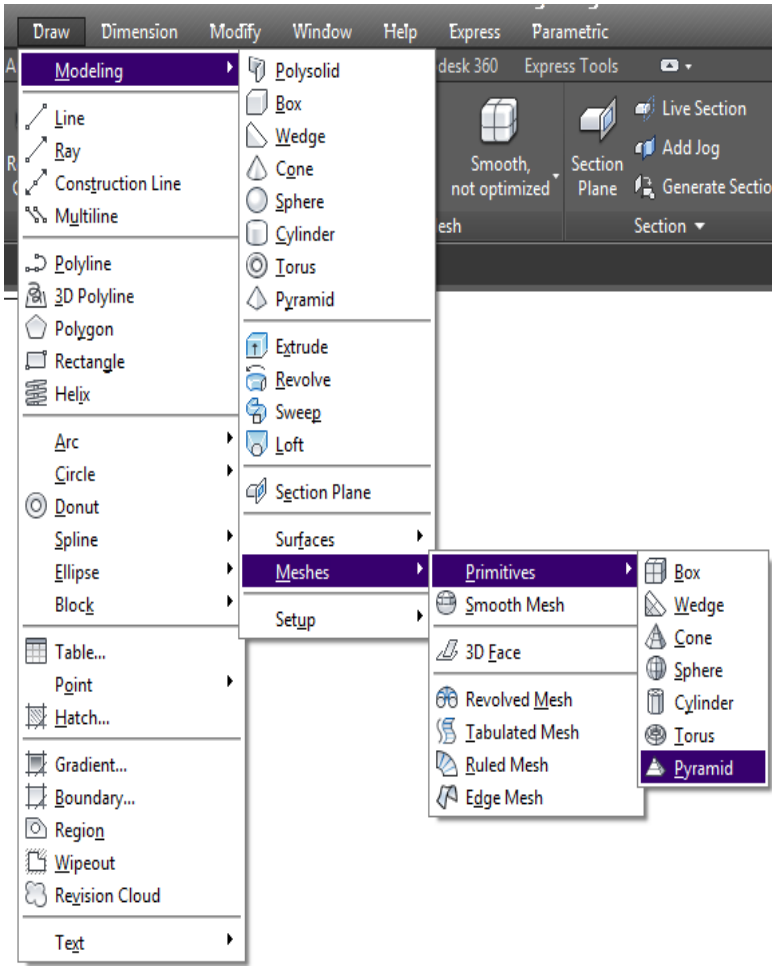
Through Ribbon

This figure describes the location of Mesh Pyramid command of 3D Surface in Mesh Tab.



Through Main Menu

Main Menu →Draw →Modeling →Meshes→Primitives →Pyramid

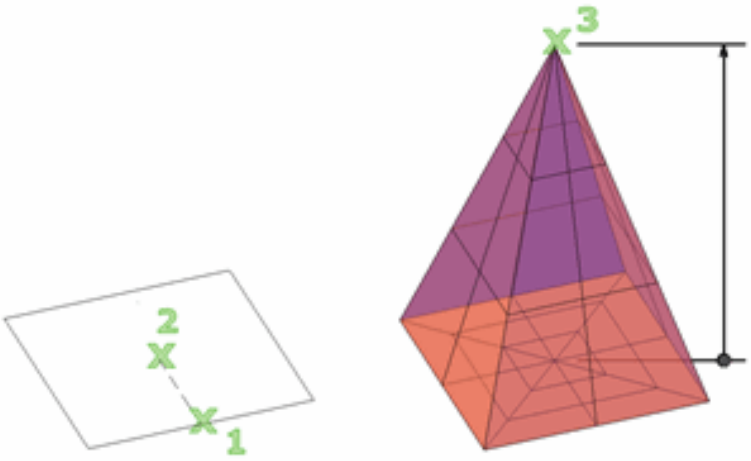


Through Short cut Key

MESH enter → P enter

APPLICATION

Type MESH at the Command Prompt →[Box/Cone/CYlinder/Pyramid/Sphere/Wedge/Torus/SEttings] <Box>:→ P enter (for Pyramid)→ Specify center point of base→Put the base radius value→Put the Value of Height→enter



Do you know!

You can create a mesh pyramid up to 32 sides.

Mesh Pyramid Option

The Pyramid option of the MESH command provides several methods for determining the size and rotation of the mesh pyramids you create.

i. Sides

ii. Edges

iii. Frustum

iv. Axis Endpoint

v. Inscribed or circumscribed perimeter

i. Sides.

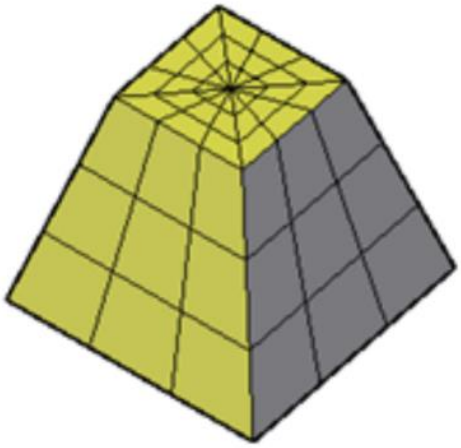
Use the Sides option to set the number of sides for the mesh pyramid.

ii. Edges.

Use the Edges option to specify the dimension of the sides at the base.

iii. Frustum

Use the Top Radius option to create a frustum, which tapers to a planar face. The frustum face is parallel to and has the same number of sides as, the base.

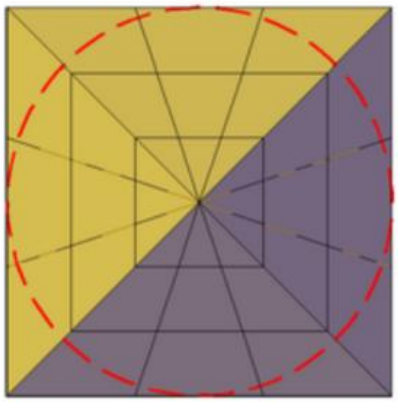


iv. Axis Endpoint

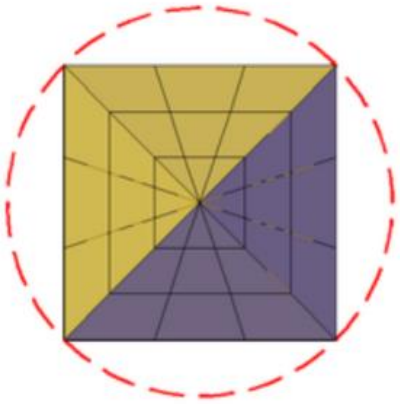
Use the Axis Endpoint option to specify the height and rotation of the pyramid. This endpoint is the top of the pyramid. The axis endpoint can be located anywhere in 3D space.

v. Inscribed or Circumscribed Perimeter.

Specify whether the pyramid base is drawn inside or outside of the radius.



Circumscribed



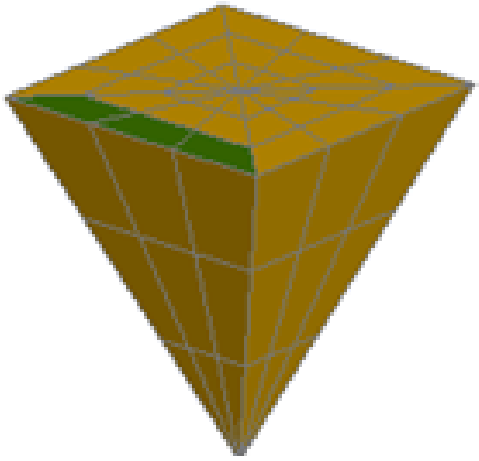
Inscribed

Tessellation Divisions of Pyramid

The following system variables are used to control the number of divisions for each dimension of a new mesh pyramid:

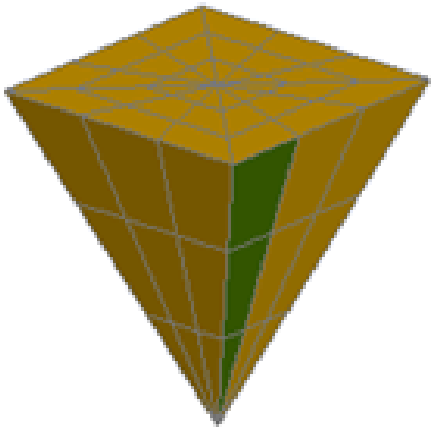
a) Length.

Set the number of divisions for the sides of the mesh pyramid base. (DIVMESHPYRLENGTH system variable)



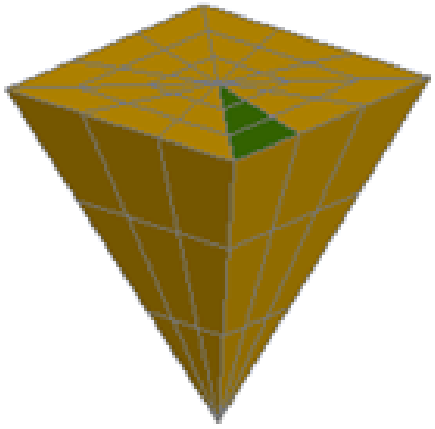
b) Height.

Set the number of divisions between the base and the top of the mesh pyramid. (DIVMESHPYRHEIGHT system variable)



c) Base.

Set the number of divisions between the perimeter and the center point of the mesh pyramid base. (DIVMESHPYRBASE system variable)



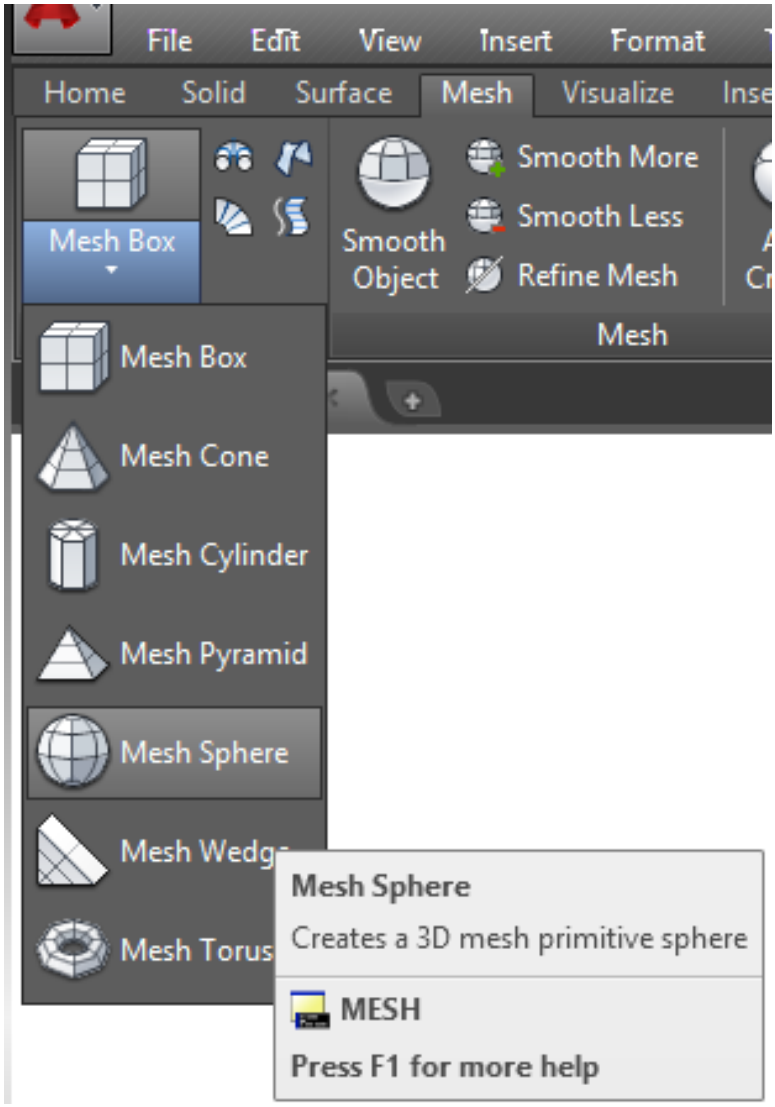
5) Mesh Sphere

You can create a sphere by specifying a center point and a point on the radius. You can control the smoothness of curved 3D solids, such as a sphere, in a Shaded or hidden visual style with the FACETRES system variable.

Mesh Sphere command of 3D Surface can be find out through following ways

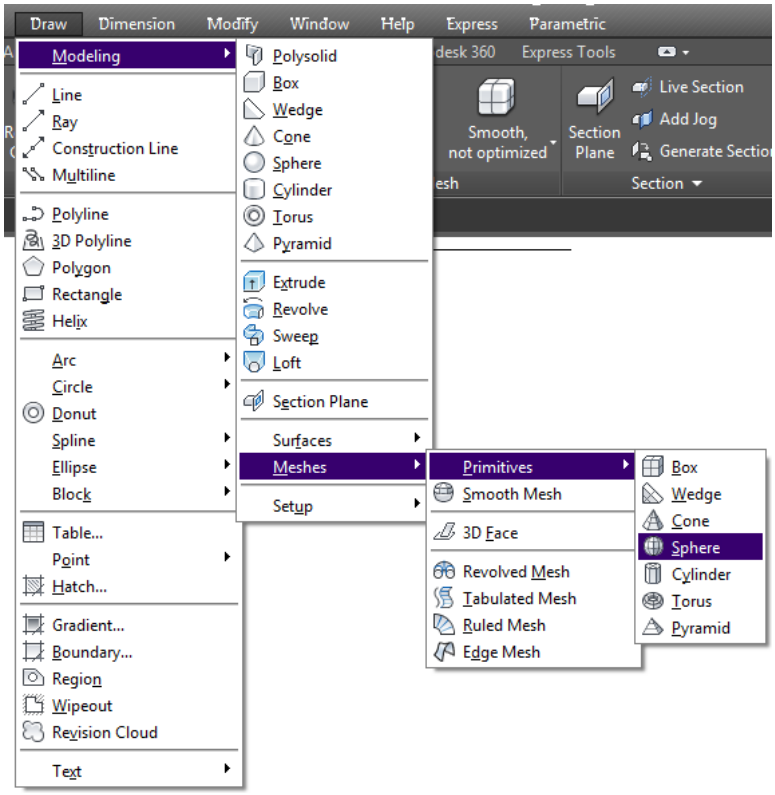
Through Ribbon

This figure describes the location of Mesh Sphere command of 3D Surface in Mesh Tab.



Through Main Menu

Main Menu →Draw →Modeling →Meshes→Primitives →Sphere

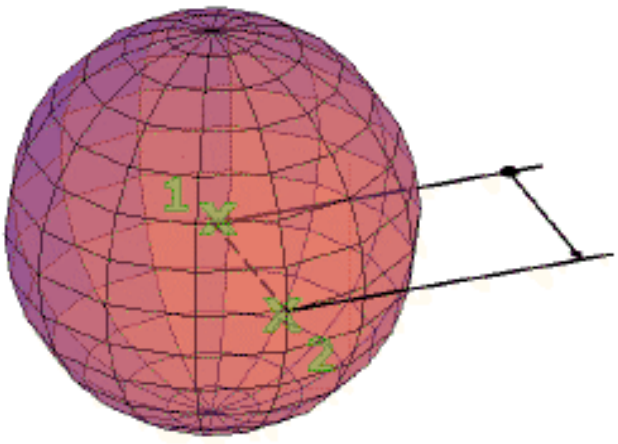


Through Short Cut Key

MESH enter → S enter

APPLICATION

Type MESH at the Command Prompt →[Box/Cone/CYlinder/ Pyramid/ Sphere/Wedge/Torus/SEttings] <Box>:→ S enter (for Sphere)→Specify center point →Put the radius value→enter



Mesh Sphere Option

i. Center point

ii. 2p (Two Point)

iii. 3p (Three Point)

iv. TTR(Tangent, Tangent, Radius)

i. Center Point

Specifies the center point of the sphere. When you specify the center point, the sphere is positioned so that its central axis is parallel to the Z axis of the current user coordinate system (UCS).Latitudinal lines are parallel to the XY plane.

- **Radius** Defines the radius of the sphere.
- **Diameter** Defines the diameter of the sphere.

ii. 3P (Three Points)

Defines the circumference of the sphere by specifying three points anywhere in 3D space. The three specified points also define the plane of the circumference.

iii. 2P (Two Points)

Defines the circumference of the sphere by specifying two points anywhere in 3D space. The plane of the circumference is defined by the Z value of the first point.

iv. TTR (Tangent, Tangent, Radius)

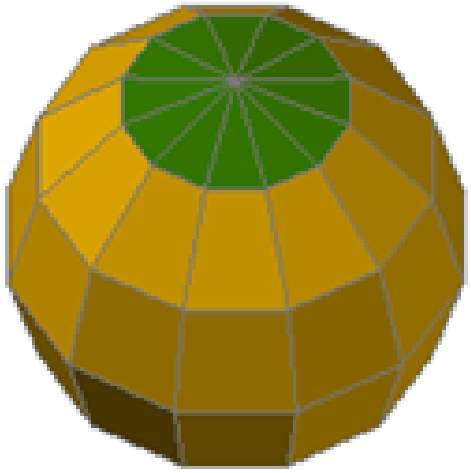
Defines the sphere with a specified radius tangent to two objects. The specified tangency points are projected onto the current UCS.

Tessellation Divisions of Sphere

The following system variables are used to control the number of divisions for each dimension of a new mesh sphere

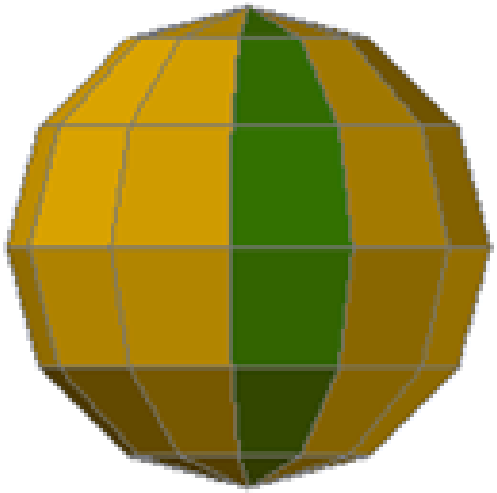
a) Axis.

Set the number of radial divisions around the axis endpoint of the mesh sphere. (DIVMESHSPHEREAXIS system variable)



b) Height.

Set the number of divisions between the two axis endpoints of the mesh sphere. (DIVMESHSPHEREHEIGHT)



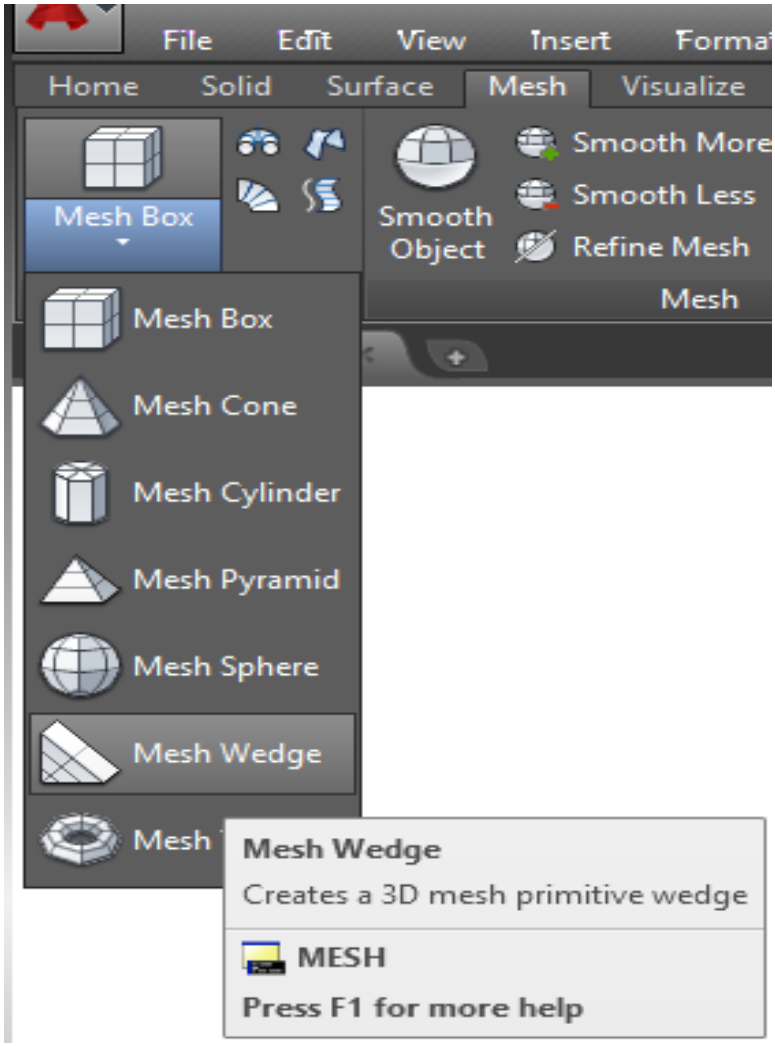
6) Mesh Wedge

Creates a mesh wedge with rectangular or cubical faces. The base of the wedge is drawn parallel to the XY plane of the current UCS with the sloped face opposite the first corner. The height of the wedge is parallel to the Z axis.

Mesh Wedge command of 3D Surface can be find out through following methods:

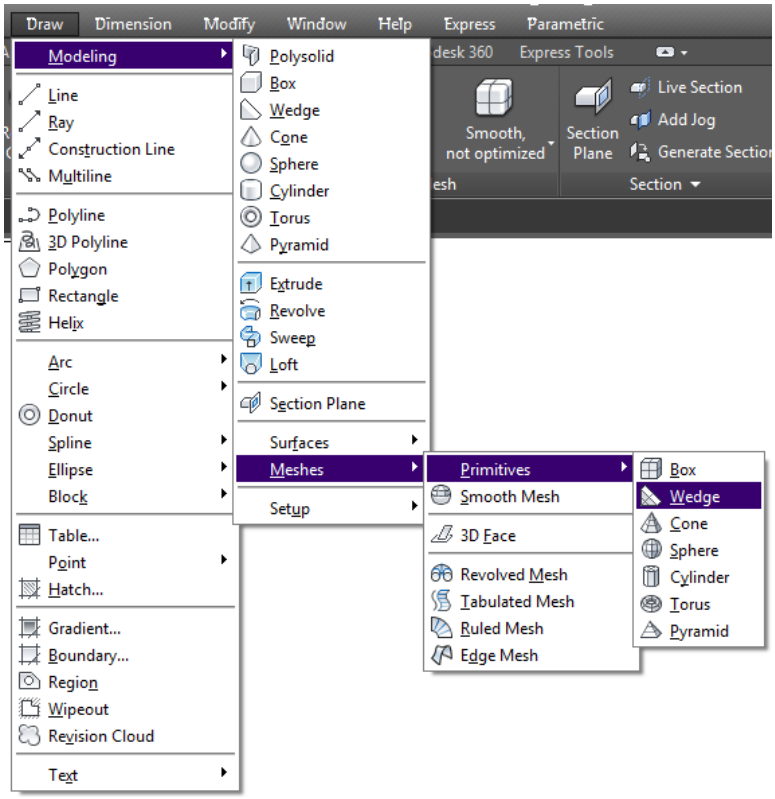
Through Ribbon

This figure describes the location of Mesh Wedge command of 3D Surface in Mesh Tab.



Through Main Menu

Main Menu →Draw →Modeling →Meshes→Primitives →Wedge

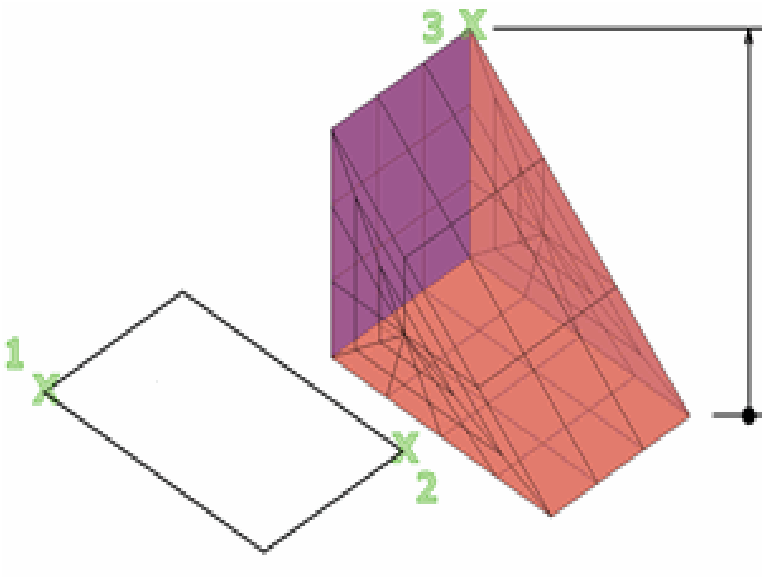


Through Short cut Key

MESH enter → W enter

APPLICATION

Type MESH at the Command Prompt →[Box/Cone/CYlinder/ Pyramid/Sphere/Wedge/Torus/SEttings] <Box>:→ W enter (for Wedge)→ Specify Start point→Put the Value of @ X and Y Co-ordinate→Put the Value of Height→enter



Mesh Wedge Option

The Wedge option of the MESH command provides several methods for determining the size and rotation of the mesh wedges you create.

i. **Center**

ii. **Cube**

iii. **Length**

iv. **2 Point (With Height)**

i. Center

Create the wedge by using a specified center point

ii. Cube

Set all edges of the mesh wedge base to be of equal length.

iii. Length

Creates a wedge with length, width, and height values you specify. The length corresponds to the X axis, the width to the Y axis, and the height to the Z axis. If you pick a point to specify the length, you also specify the rotation in the XY plane.

iv. 2 point (with Height)

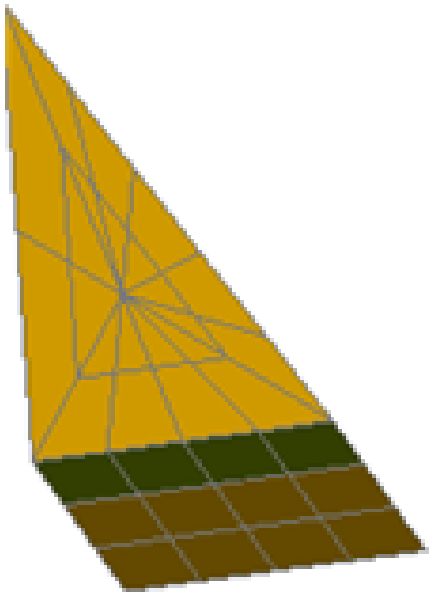
Specifies that the height of the wedge is the distance between the two specified points.

Tessellation Divisions of Wedge

The following system variables are used to control the number of divisions for each dimension of a new mesh wedge.

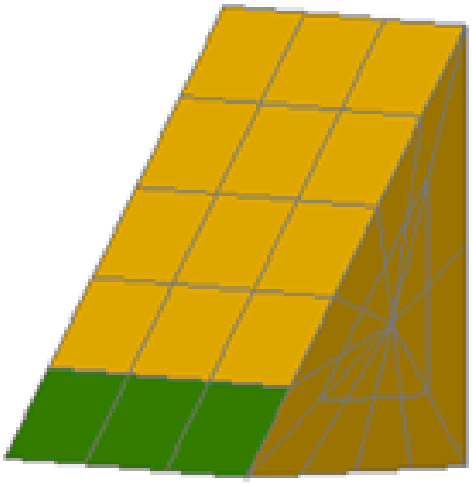
a) Length.

Set the number of divisions for the length of a mesh wedge along the X axis. (DIVMESHWEDGELENGTH system variable)



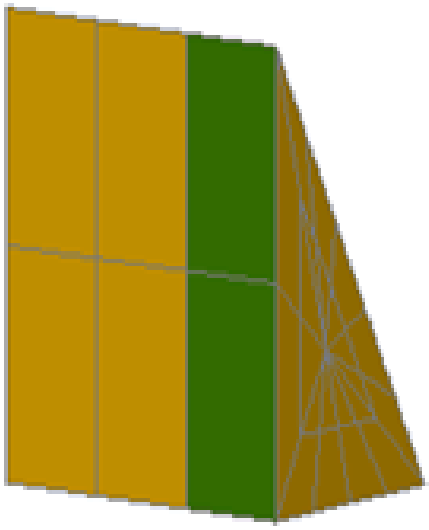
b) Width.

Set the number of divisions for the width of the mesh wedge along the Y axis. (DIVMESHWEDGEWIDTH system variable)



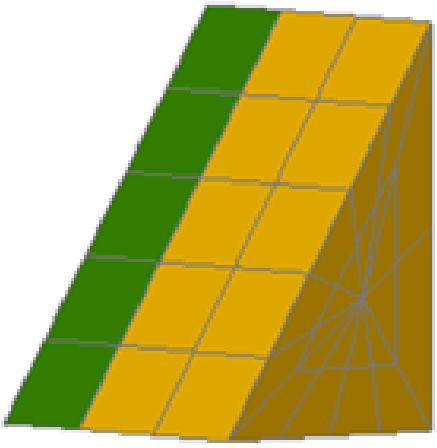
c) Height.

Set the number of divisions for the height of the mesh wedge along the Z axis. (DIVMESHWEDGEHEIGHT system variable)



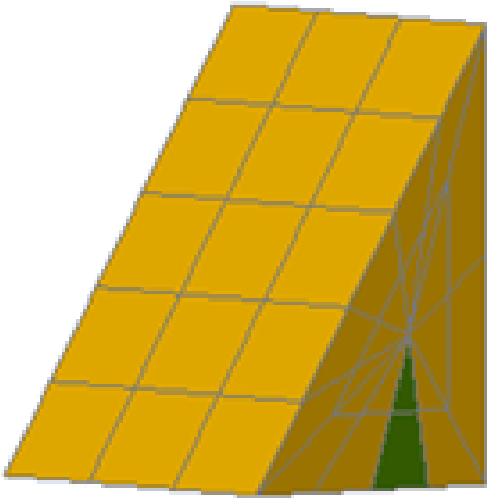
d) Slope.

Set the number of divisions in the slope that extends from the apex of the wedge to the edge of the base. (DIVMESHWEDGESLOPE system variable)



e) Base.

Set the number of divisions between the midpoints of the perimeter of the triangular side of the mesh wedge. (DIVMESHWEDGEBASE system variable)



7) Mesh Torus

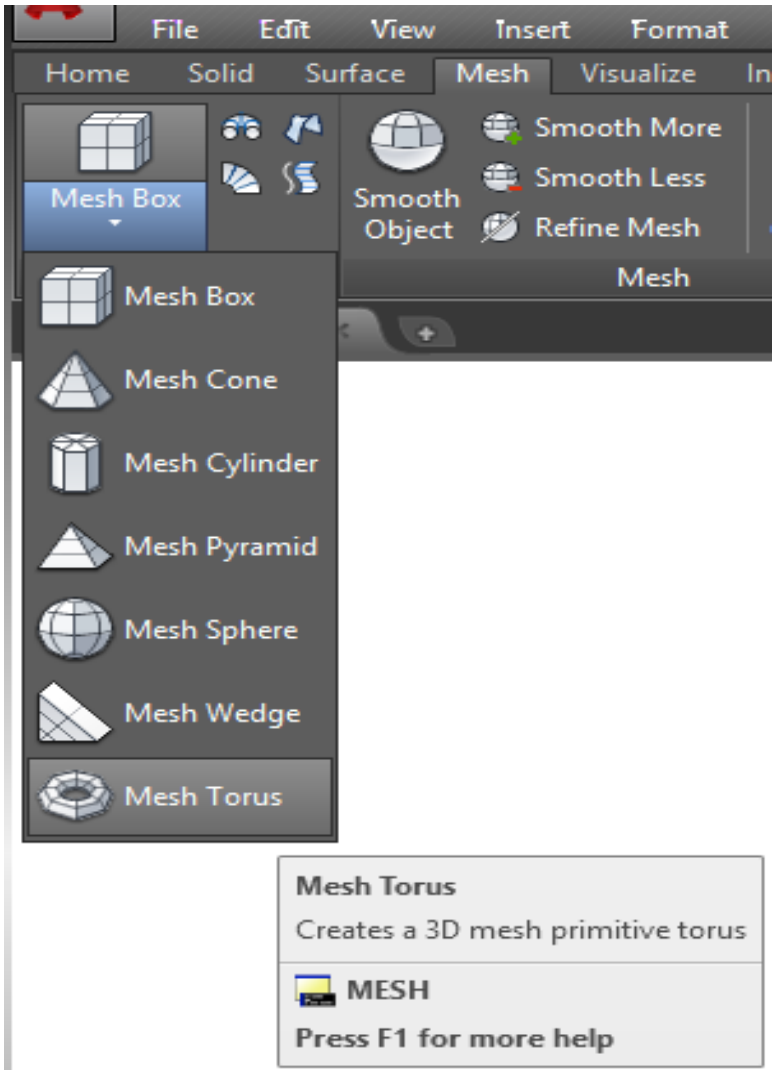
Creates a ring-shaped surface that resembles the inner tube of a tire. A mesh torus has two radius values. One value defines the tube. The other value defines the path, which is equivalent to the distance from the center of the torus to the center of the tube. By default, a torus is drawn parallel to and is bisected by the XY plane of the current UCS.

A mesh torus can be self-intersecting. A self-intersecting mesh torus has no center hole because the radius of the tube is greater than the radius of the torus.

Mesh Torus command of 3D Surface can be find out through following methods:

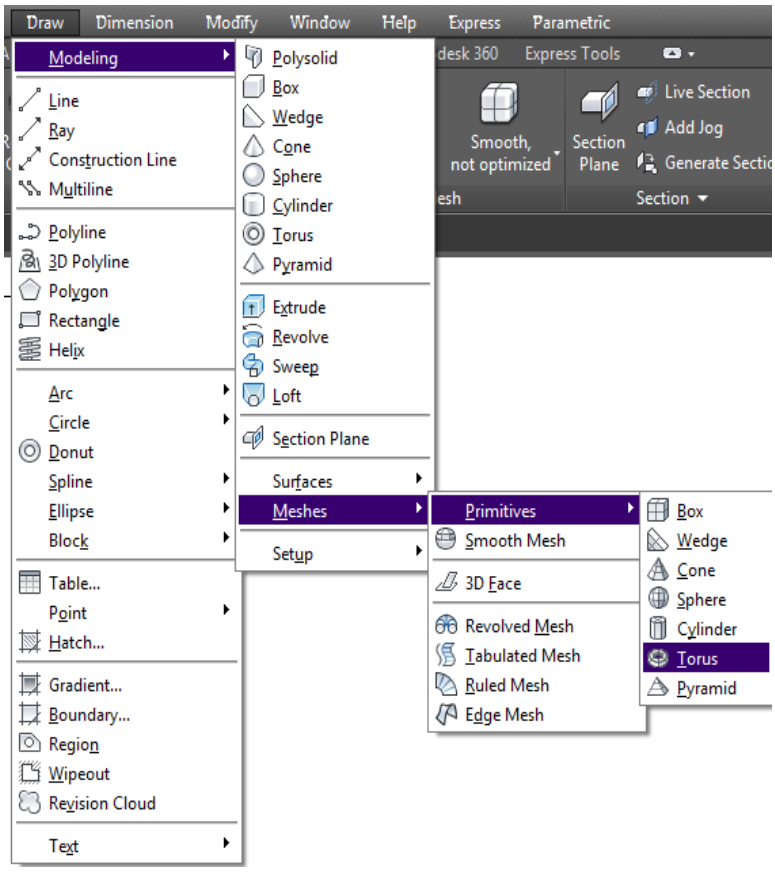
Through Ribbon

This figure describes the location of Mesh Torus command of 3D Surface in Mesh Tab.



Through Main Menu

Main Menu →Draw →Modeling →Meshes→Primitives →Torus

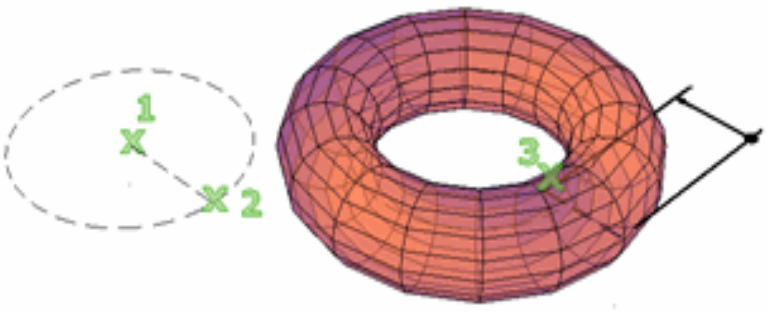


Through Short cut Key

MESH enter → T enter

APPLICATION

Type MESH at the Command Prompt →[Box/Cone/CYlinder/ Pyramid/Sphere/Wedge/Torus/SEttings] <Box>:→ T enter (for Torus)→ Specify center point →Put the radius value→Put the Tube radius Value →enter



Mesh Torus Option

The Torus option of the MESH command provides several methods for determining the size and rotation of the mesh tours you create.

i. 3P

ii. 2p

iii. TTR

iv. Radius

i. 3P (Three Points)

Use the 3P (Three Points) option to define the size of the mesh torus anywhere in 3D space. The three points also define the plane of the circumference. Use this option to rotate the mesh torus as you create it.

ii. 2P (Two Points)

Use the 2P (Two Points) option to define the size of the mesh torus anywhere in 3D space. The plane of the circumference matches the Z value of the first point.

iii. TTR (Tangent, Tangent, Radius)

Use the TTR (Tangent, Tangent, and Radius) option to define points on two objects. Depending on the specified radius distance, the path of the torus is located as near as possible to the tangent points you specify. You can set up tangency with circles, arcs, lines, and some 3D objects. The tangency points are projected into the current UCS. The appearance of tangency is affected by the current level of smoothness.

iv. Radius

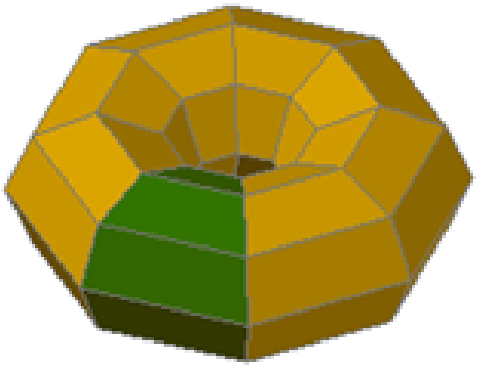
Defines the radius of the torus: the distance from the center of the torus to the center of the tube. A negative radius creates a solid shaped like an American football.

Tessellation Divisions of Torus

The following system variables are used to control the number of divisions for each dimension of a new mesh torus:

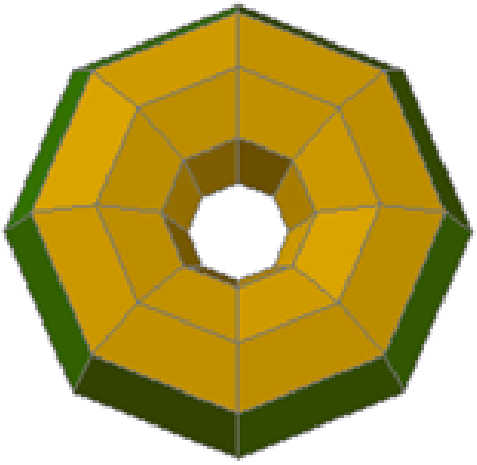
a) Radius.

Set the number of divisions around the circumference of the profile that forms the tube of the torus. (DIVMESHTORUSSECTION system variable)



b) Sweep Path

Set the number of divisions around the circumference of the path that is swept by the profile.(DIVMESHTORUSPATH system variable)



8) Mesh Tessellation settings

Modifies the smoothness and tessellation values for the new mesh object.

Level of smoothness

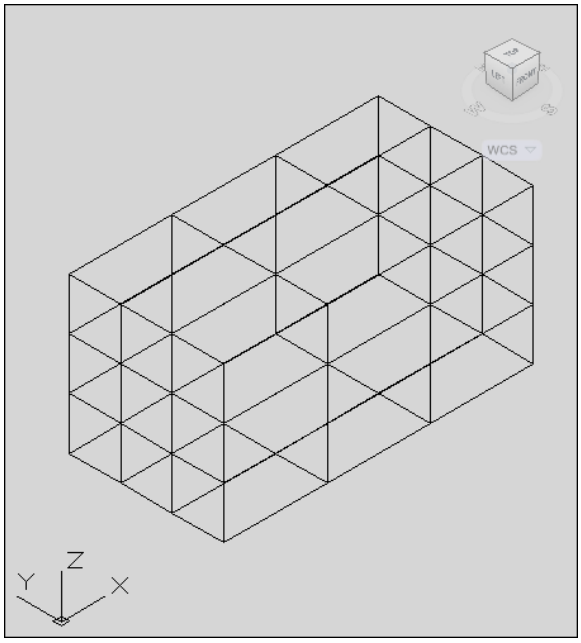
Set the initial degree of smoothness, or roundness, to be applied to the mesh. Enter a positive integer up to 4 for increased degrees of smoothness. The value is retained for the current drawing session. This smoothness value is limited by the value of SMOOTHMESHMAXLEV.

Tessellation

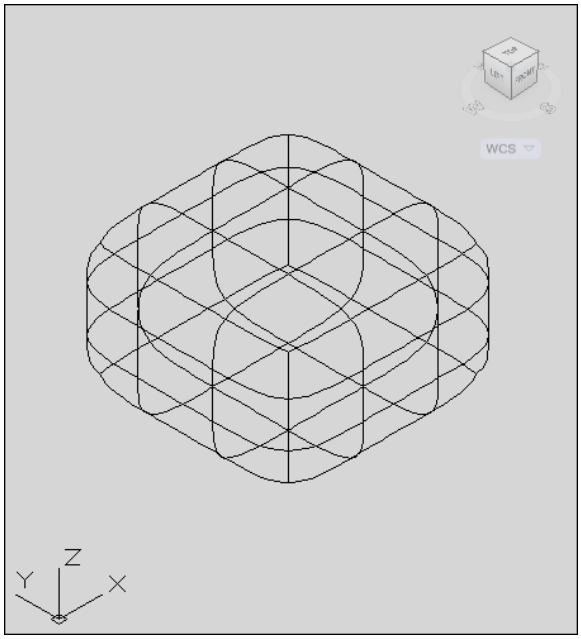
Opens the Mesh Primitive Options dialog box, where you can update tessellation values (the number of faces) for each dimension of a mesh primitive option.

APPLICATION

Type MESH at the Command Prompt →[Box/Cone/CYlinder/ Pyramid/Sphere/Wedge/Torus/SEttings] <Box>:→ SE enter (for Setting)→ change the smoothness level Put the Value (0-4)→ enter



Smoothness Level 0



Smoothness Level 3

Remember

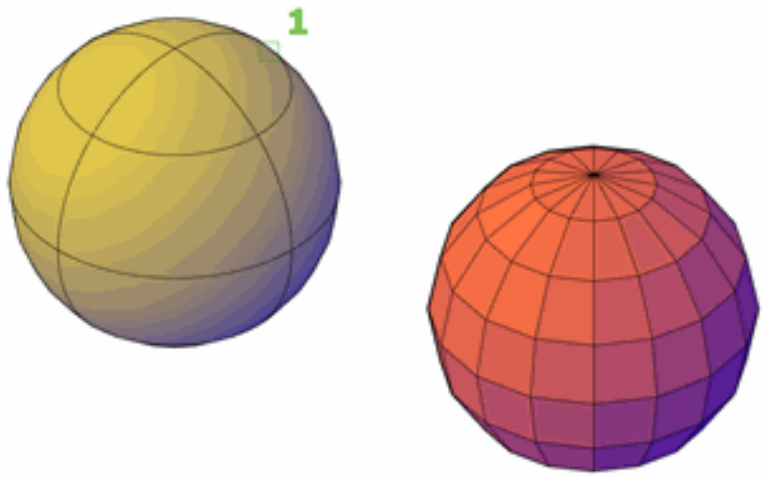
By default the value of smooth will be 0, you can adjust it value up to 4

3. Smooth Meshes

Convert 3D objects such as polygon meshes, surfaces, and solids to mesh objects. Take advantage of the detailed modeling capabilities of 3D mesh by converting objects such as 3D solids and surfaces to mesh.

Do you know!

You can also convert mesh objects into 3D surfaces. To convert mesh objects to 3D surfaces or solids, use **CONVTOSOLID** or **CONVTOSURFACE** commands.



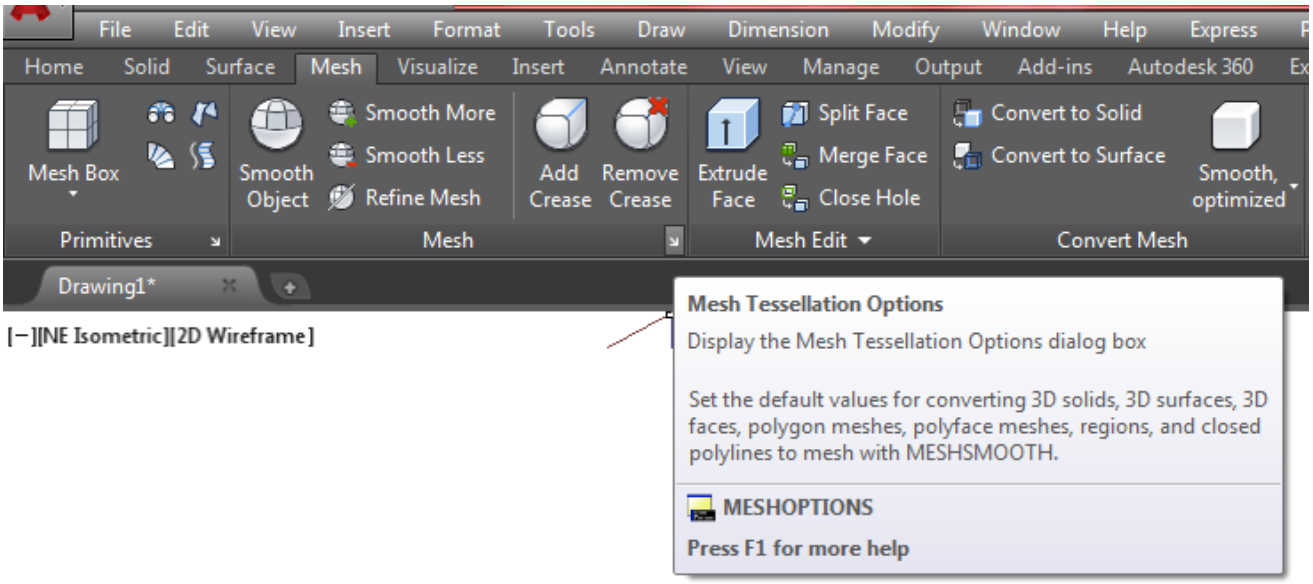
Use this method to convert 3D faces (3DFACE) and legacy polygonal and poly face meshes you can also convert 2D objects such as regions and closed polylines.

The default mesh settings are defined in the Mesh Tessellation Options dialog box. The level of smoothness upon conversion depends on the mesh typesetting in this dialog box. If the mesh type is not set to be optimized, the converted object is not smoothed.

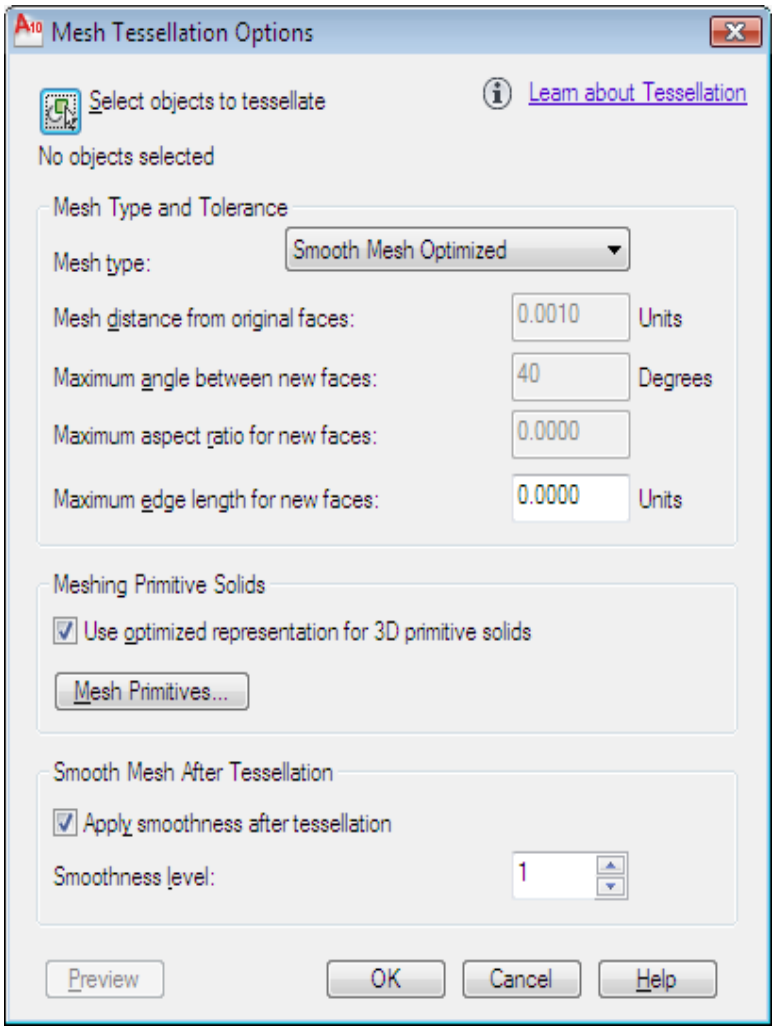
Show Mesh Tessellation Options dialog box

Click the Mesh tab→Click the Mesh panel

→Show Mesh tessellation Options Dialog Box



Mesh Tessellation Options dialog box



The defaults that you set in the Mesh Tessellation Options dialog box control the appearance of objects that you convert to 3D mesh objects. For a complete list of objects that can be converted to mesh, see the **MESHSMOOTH** command.

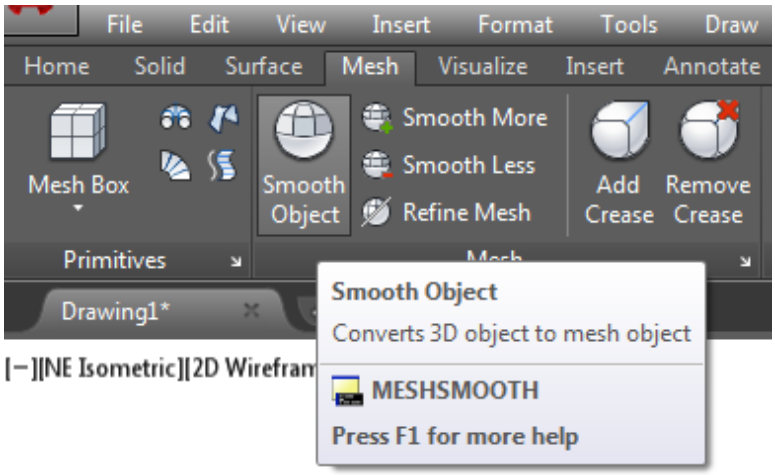
Objects That Can Be Converted to Mesh

- 3D Solid
- 3D Surface
- 3D Faces
- Poly face and polygon meshes (legacy)
- Regions object
- Closed polylines

Smooth Object command of 3D Surface can be find out through following methods.

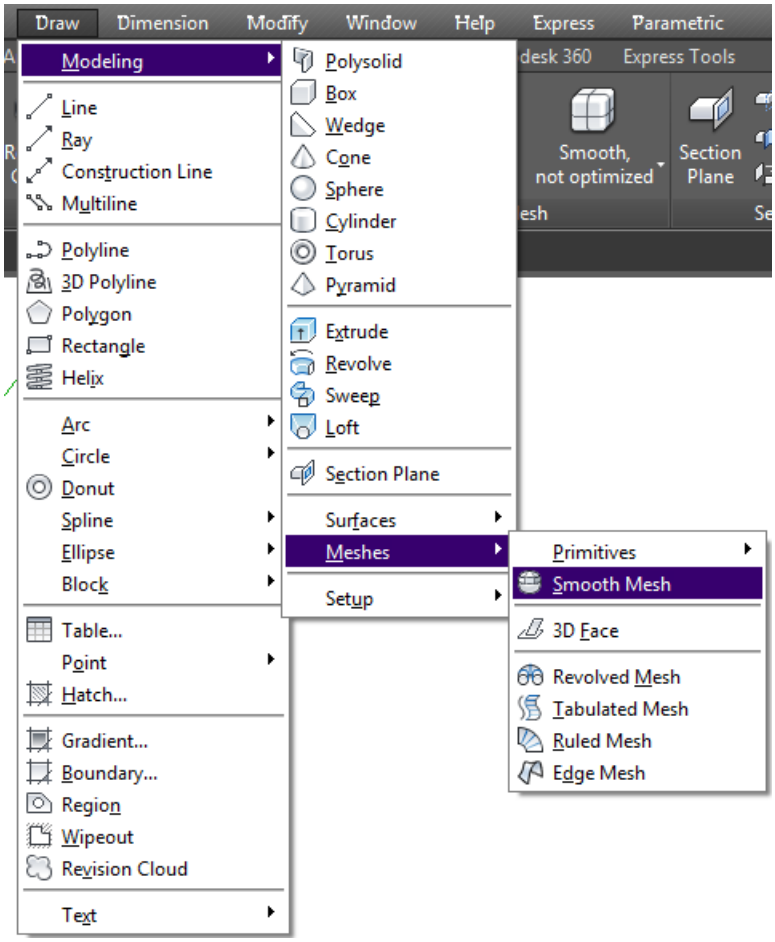
Through Ribbon

This figure describes the location of Smooth Object Command for 3D Surface in Mesh Tab.



Through Main Menu

Main Menu →Draw →Modeling →Meshes→Smooth Mesh

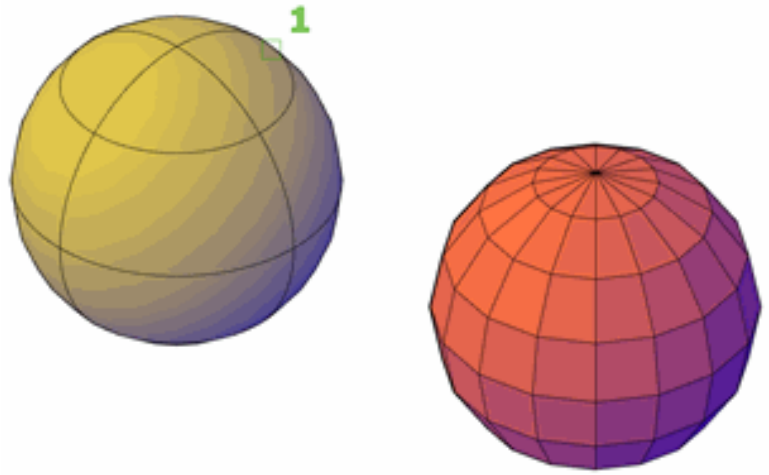


Through Short cut Key

MESHSMOOTH → enter

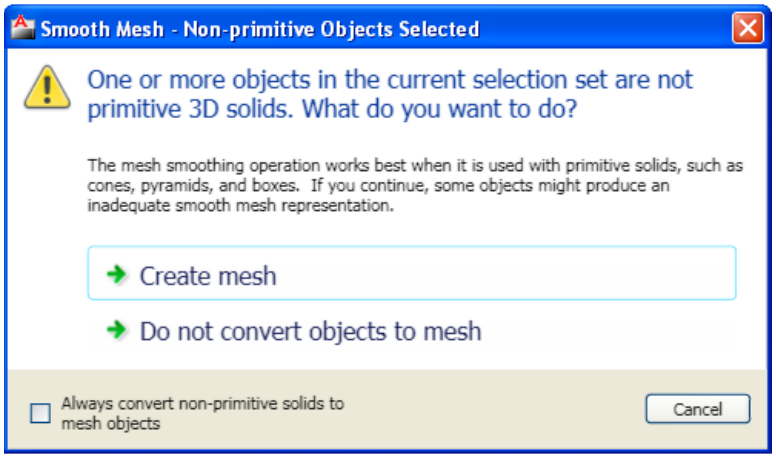
APPLICATION (Smooth Mesh for Primitive Objects)

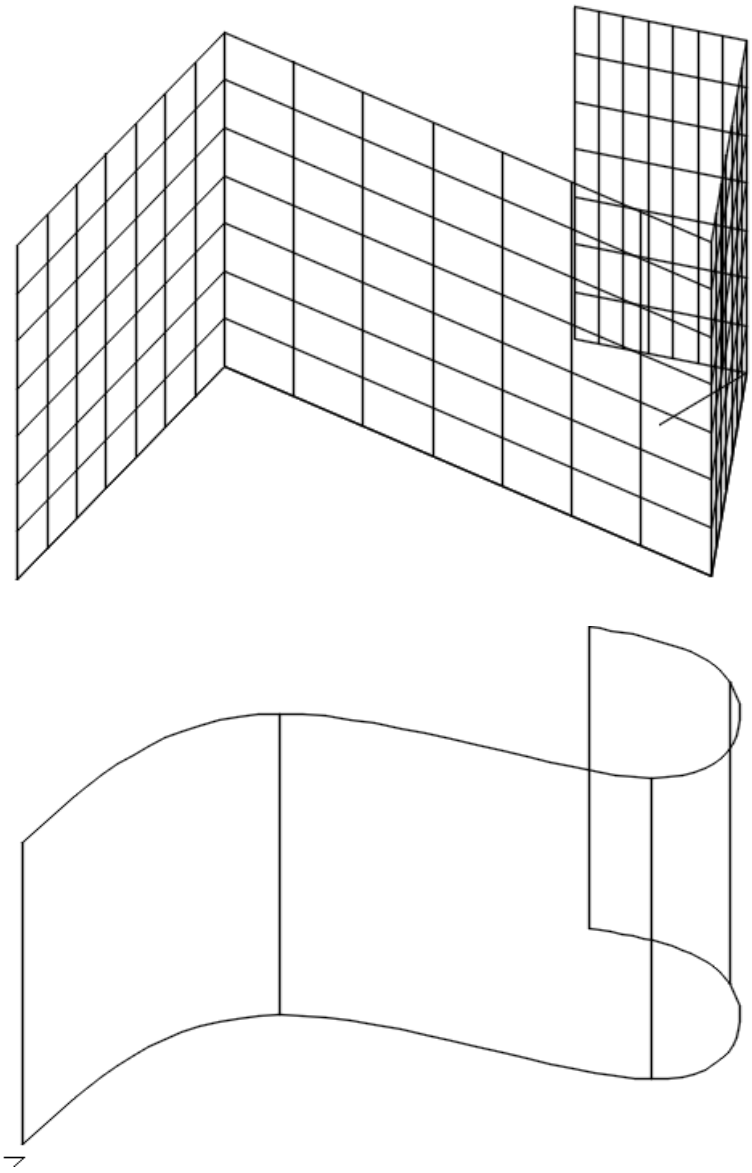
Type MESHSMOOTH at the Command Prompt →Select Object to convert → enter



APPLICATION (Smooth Mesh for Non Primitive Objects)

Select Non Primitive Object →enter→ Show this Window→Click Create Mesh→enter

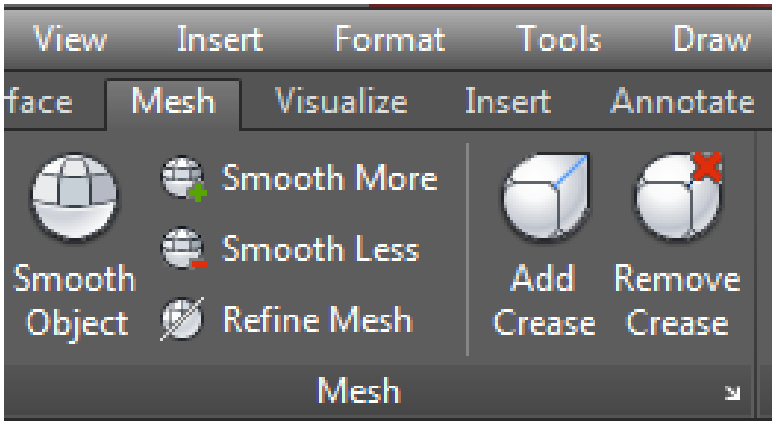




2D Polyline before MESHSMOOTH

4. Smooth and Refine Meshes

The following commands control the smoothness of mesh objects.



Mesh Smooth More

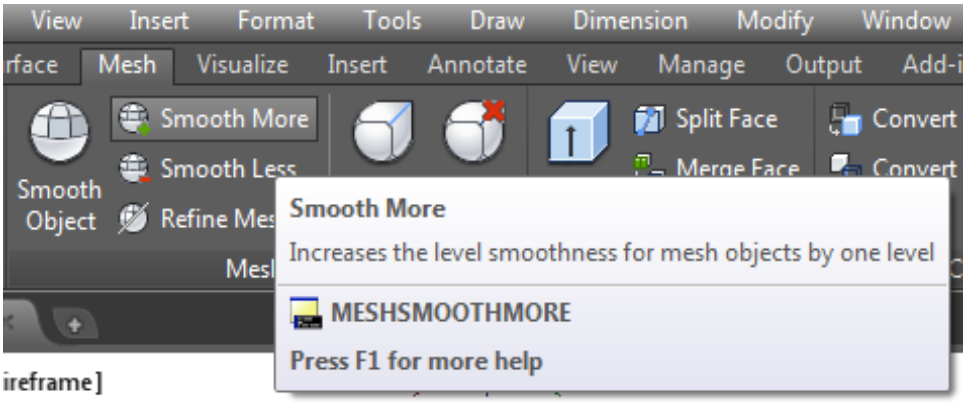
Increase the level of smoothness for mesh objects by one level. Smoothing increases the numbers of facets in the mesh, resulting in a rounder object. Facets are the underlying components of each mesh face. You can increase the smoothness level up to the value of **SMOOTHMESHMAXLEV** as long as the number of faces does not exceed the value in the **SMOOTHMESHMAXFACE** system variable. If you select multiple objects with differing levels of smoothness, their respective levels are increased by one.

You can only smooth mesh objects. However, you have the option of converting some types of objects to mesh during the smoothing operation. You can also filter out ineligible objects that you do not want to convert. For a list of objects that can be converted, see **MESHSMOOTH**.

Mesh Smooth More command of 3D Surface can be find out through following methods.

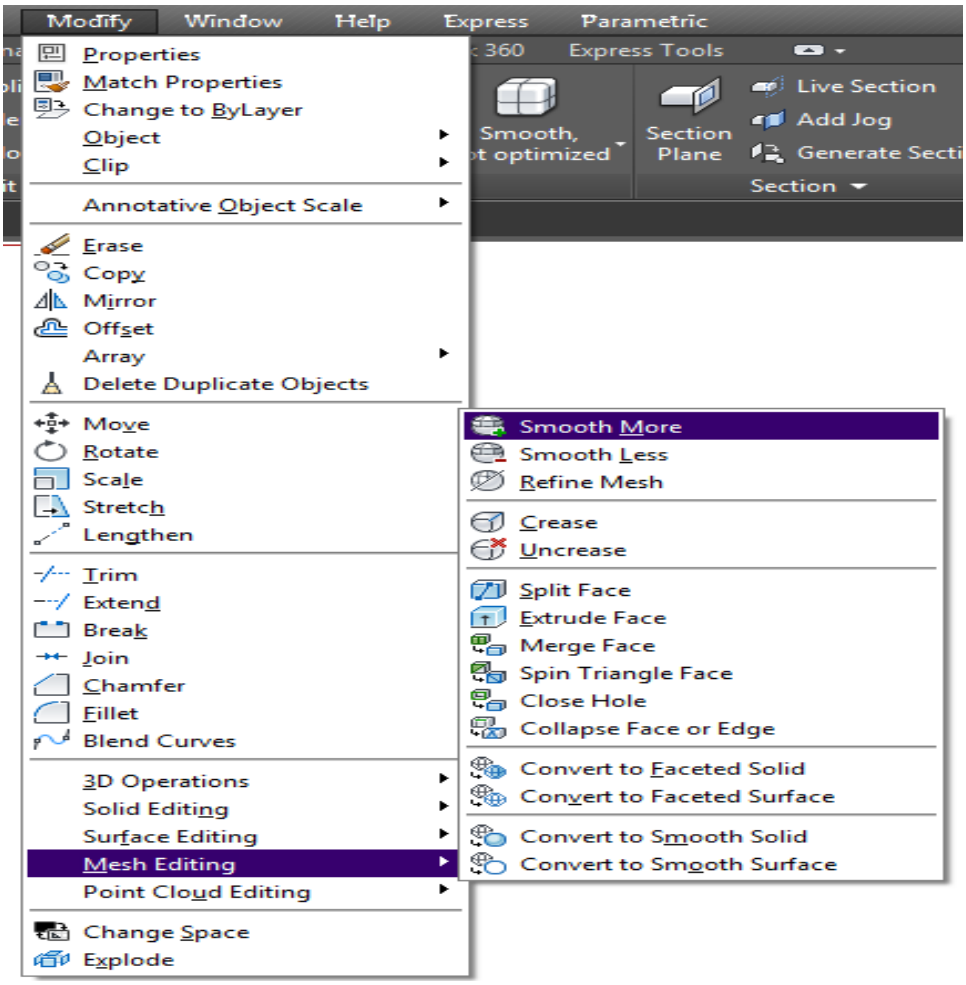
Through Ribbon

This figure describes the location of Mesh Smooth More command of 3D Surface in Mesh Tab.



Through Main Menu

Main Menu →Modify →Mesh Editing →Smooth More



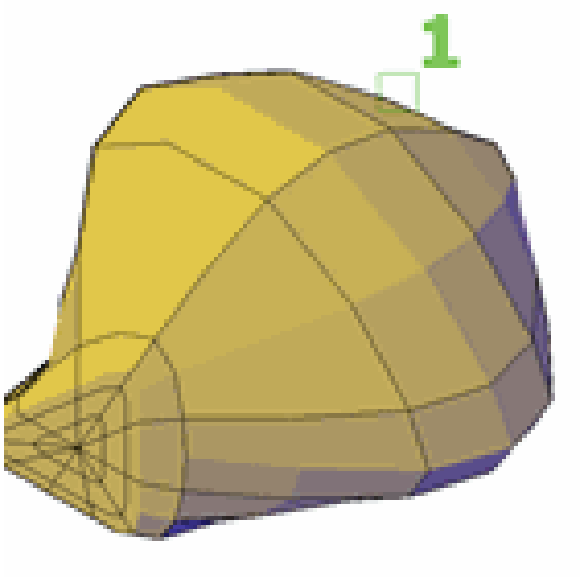
Through Short cut Key

MESHSMOOTHMORE → enter

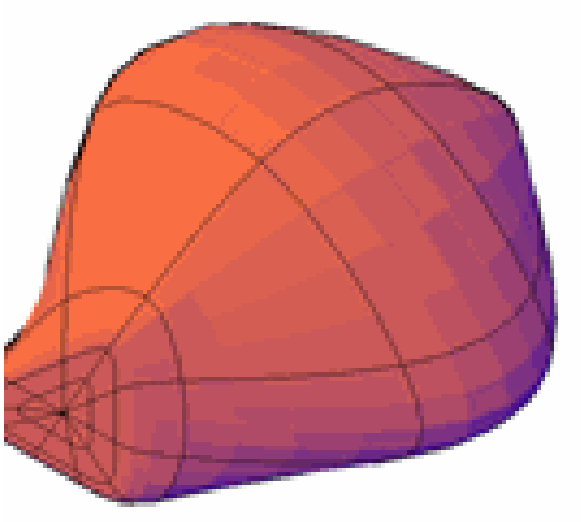
APPLICATION

Type MESHSMOOTHMORE at the Command Prompt →Select Object to convert → enter

Mesh before Smooth Increase



Mesh after Smooth Increase



Mesh Smooth Less

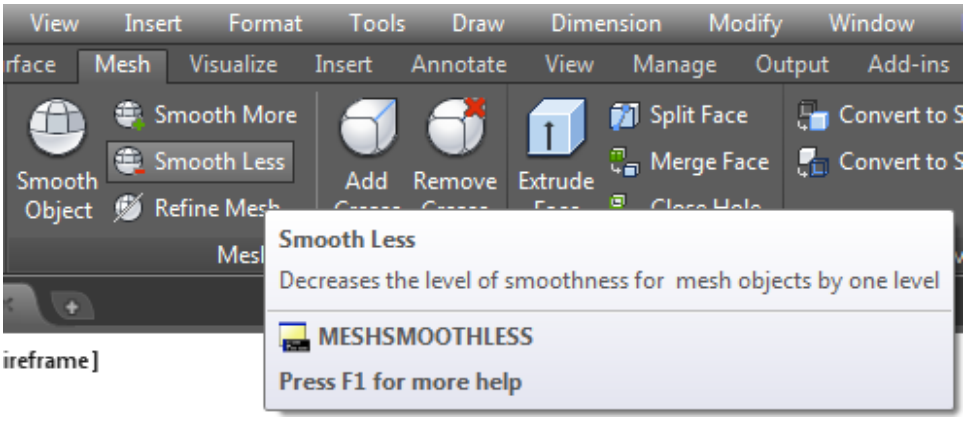
Decrease the smoothness of selected mesh objects by one level. You can only decrease the smoothness of objects whose level of smoothness is 1 or higher. You cannot decrease the smoothness level of objects that have been refined.

You can decrease the smoothness of multiple objects. If the selected objects have different levels of smoothness, their respective levels are decreased by one. If the selection set contains ineligible objects, those objects can be filtered out.

Mesh Smooth Less command of 3D Surface can be find out through following methods.

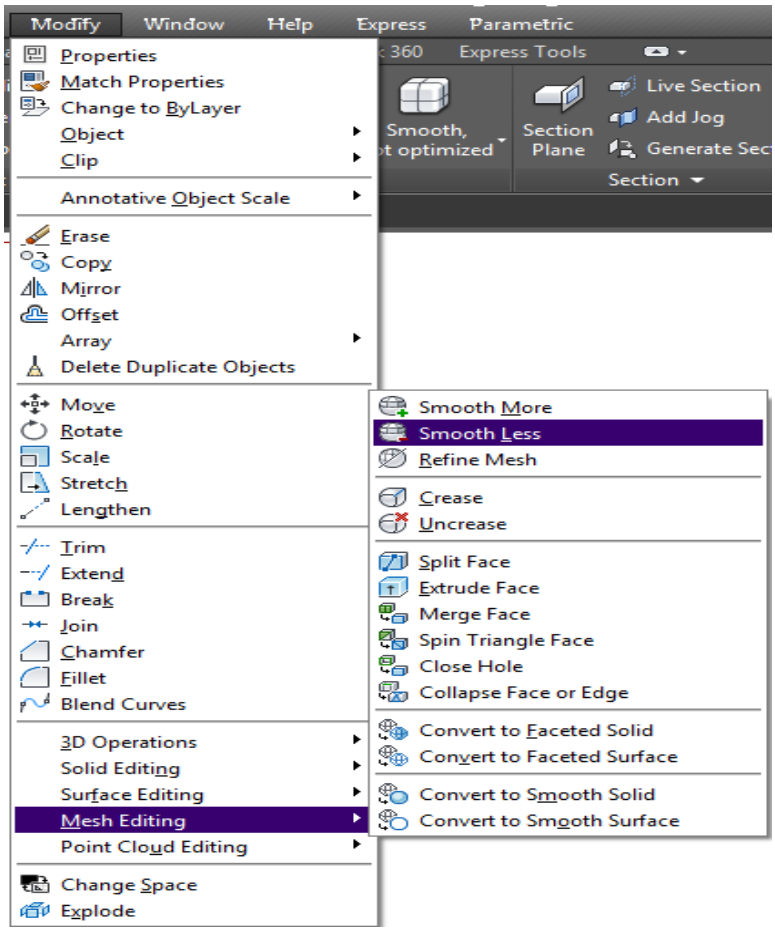
Through Ribbon

This figure describe the location of Mesh Smooth Less Command of 3D Surface in Mesh Tab



Through Main Menu

Main Menu →Modify →Mesh Editing →Smooth Less



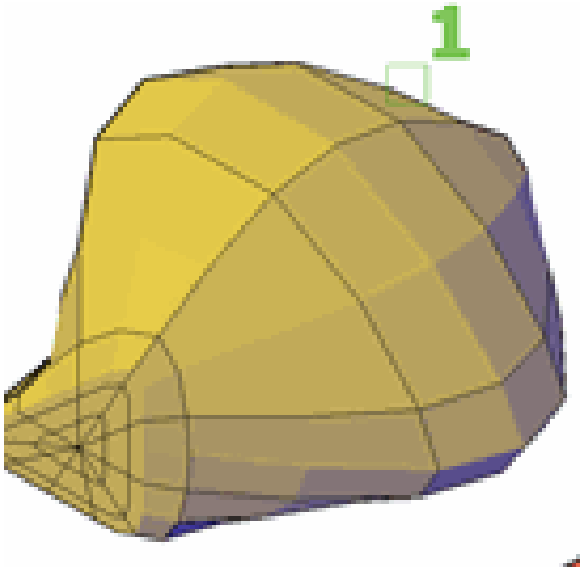
Through Short Cut Key

MESHSMOOTHLESS → enter

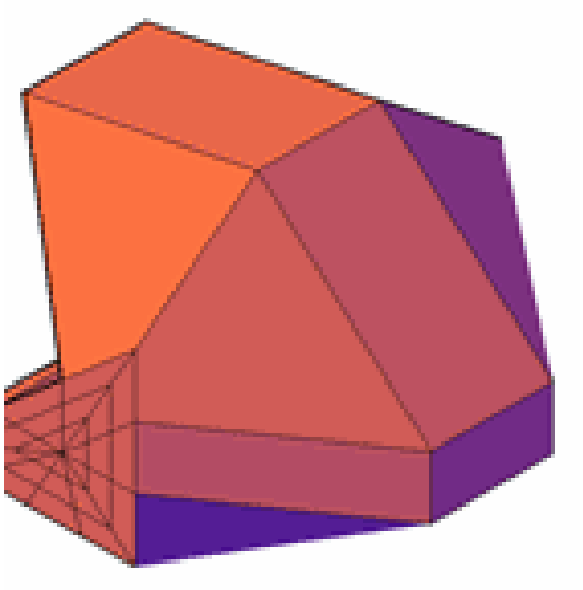
APPLICATION

Type MESHSMOOTHLESS at the Command Prompt →Select Object to convert → enter.

Mesh before Smooth Decrease



Mesh after Smooth Decrease



Mesh Refine

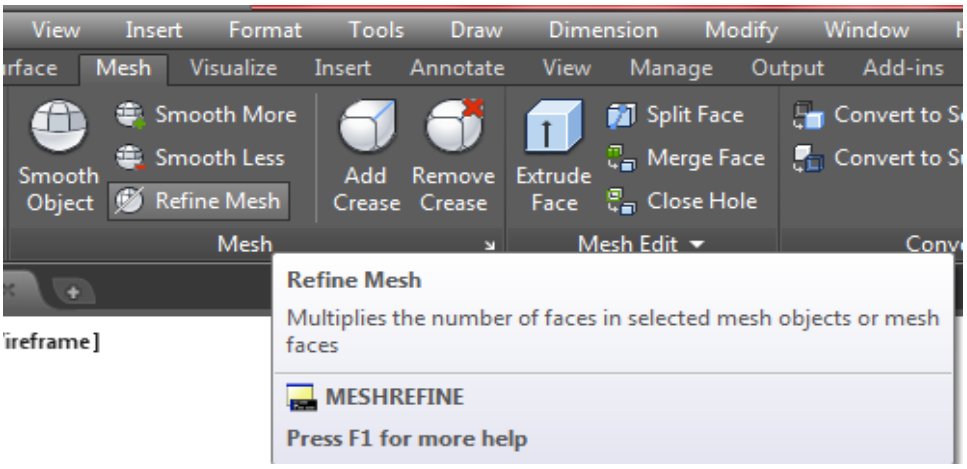
Multiple the number of faces in selected mesh objects or faces. Refining a mesh object increases the number of editable faces, providing additional control over fine modeling details. To preserve program memory, you can refine specific faces instead of the entire object.

Refining an object resets the smoothing level that was assigned to the object to 0 (zero). This level of smoothness becomes the new baseline for the object. That is, the smoothness level can no longer be decreased beyond that level. Refining a sub object does not reset the smoothing level.

Mesh Refine Command of 3D Surface can be find out through following methods.

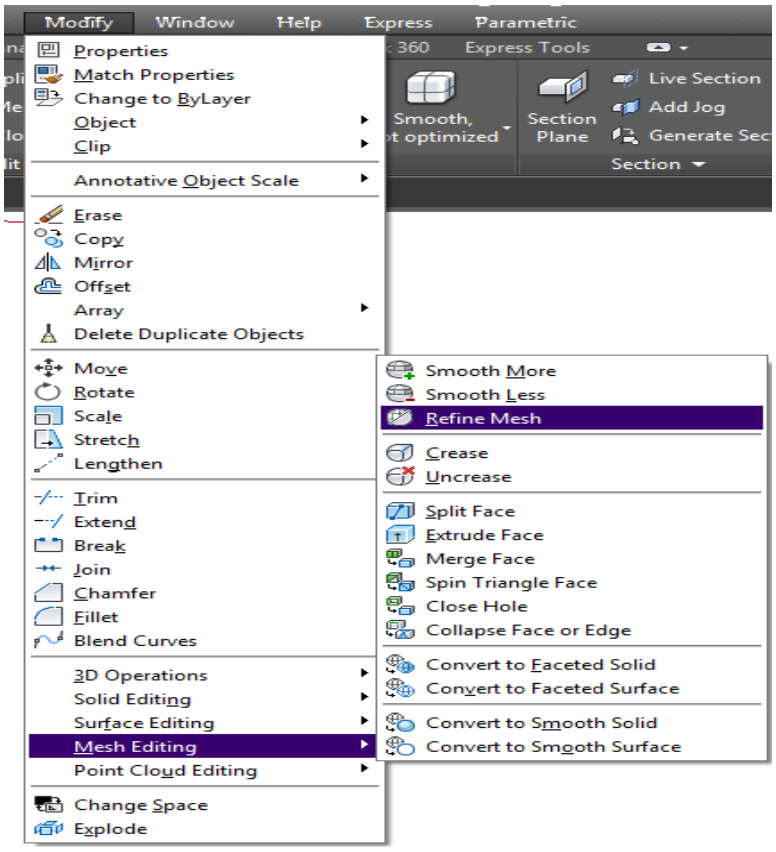
Through Ribbon

This figure describes the location of Mesh Refine Command of 3D Surface in Mesh Tab.



Through Main Menu

Main Menu →Modify →Mesh Editing →Refine Mesh



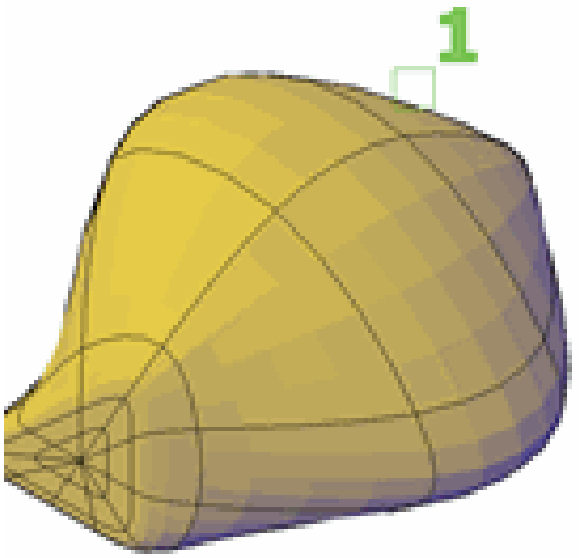
Through Short Cut Key

MESHREFINE → enter

APPLICATION

Type MESHREFINE at the Command Prompt →Select Object to convert → enter

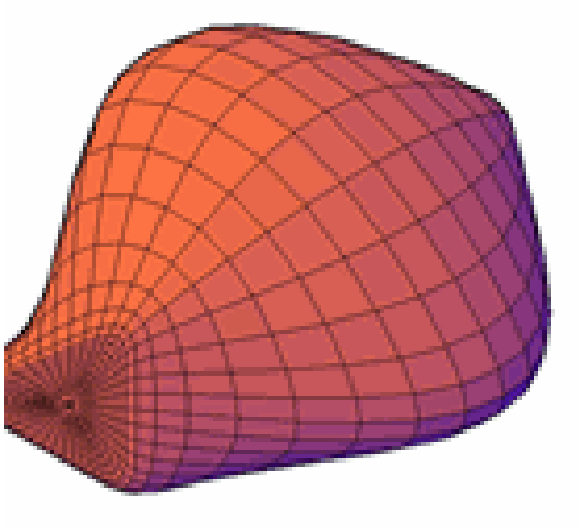
Mesh Refine Before



Remember

To refine a Mesh object or face sub objects:

Mesh Refine After



5. Add and Remove Mesh Creases

Mesh Crease

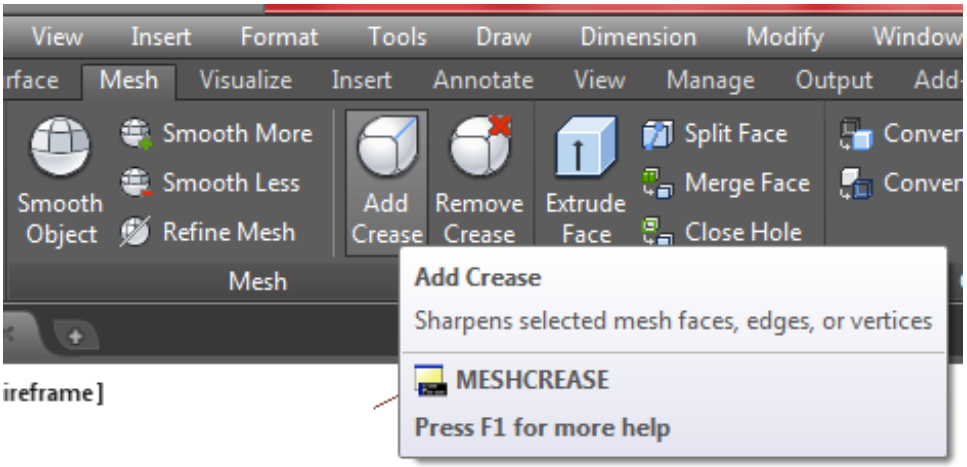
You can sharpen, or crease, the edges of mesh objects. Creasing deforms mesh faces and edges that are adjacent to the selected sub object. Creases added to mesh that have no smoothness are not apparent until the mesh is smoothed.

Sharpen the edges of selected mesh sub objects. You can sharpen, or crease, the edges of mesh objects. Creasing deforms mesh faces and edges that are adjacent to the selected sub object. You can also apply creases to mesh sub objects by changing the crease type and crease level in the Properties palette.

Mesh Crease Command of 3D Surface can be find out through following methods.

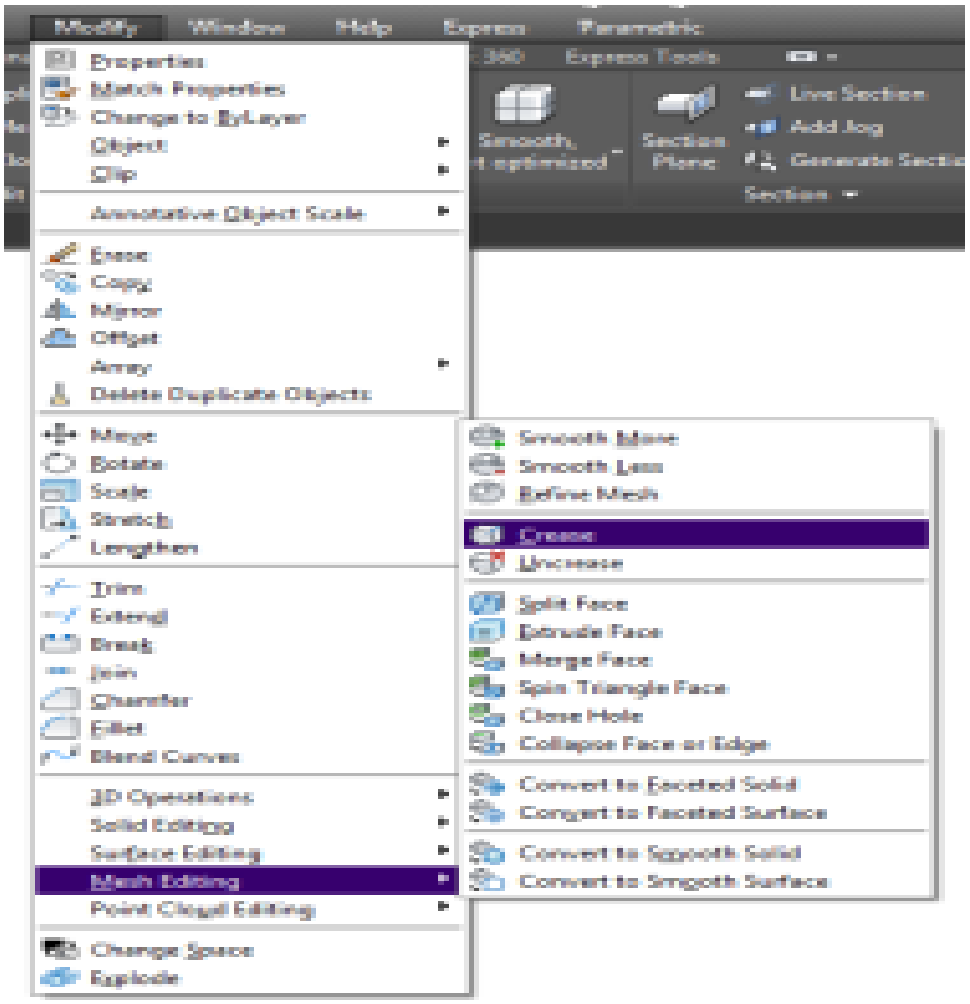
Through Ribbon

This figure describes the location of Mesh Crease Command of 3D Surface in Mesh Tab.



Through Main Menu

Main Menu →Modify →Mesh Editing →Crease



Remember

Select Sub object with the help of CTRL Key

- **Through Short Cut Key**
  MESHCREASE → enter

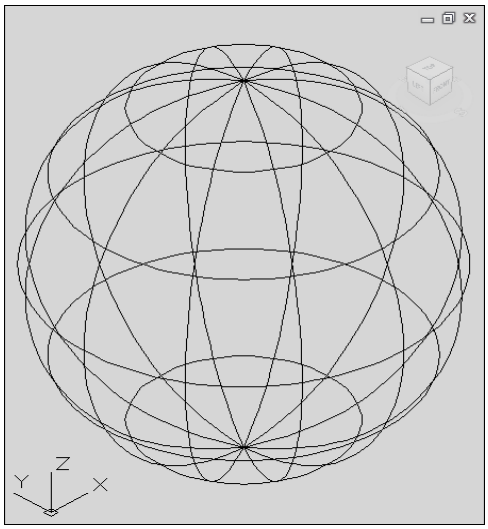
- **APPLICATION**

  Type MESHCREASE at the Command Prompt →Select Sub objects to convert → enter

Before Crease Object

Do you know!

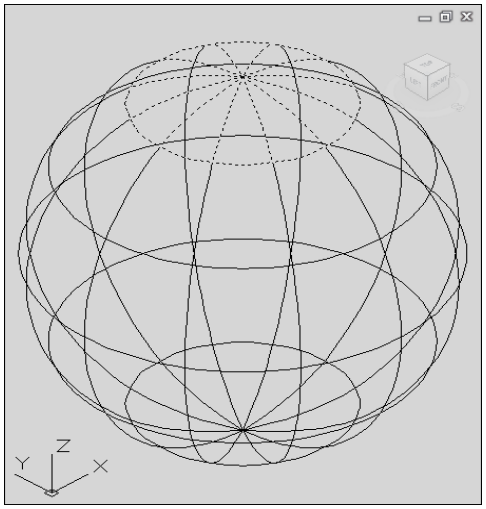
Crease value
Enter a value of 0 to remove an existing crease.
Sets highest smoothing level at which the crease is retained. If the smoothing level exceeds this value, the crease is remains smoothed.



Selected Sub Objects

Remember

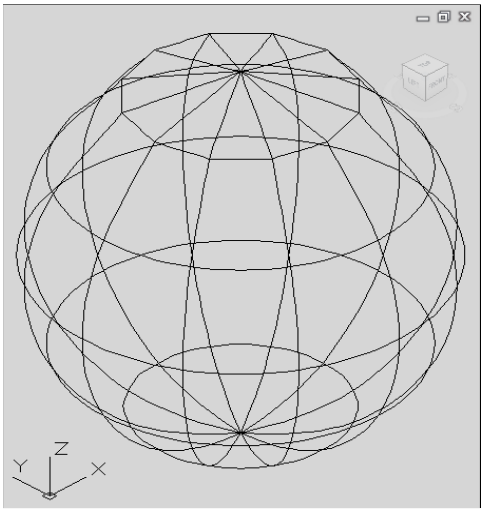
Press Shift+click to remove a sub object from the selection set.



After Crease Object

Remember

The crease is always retained, even if the object or sub object is smoothed or refined. A crease value of -1 is the same as always.



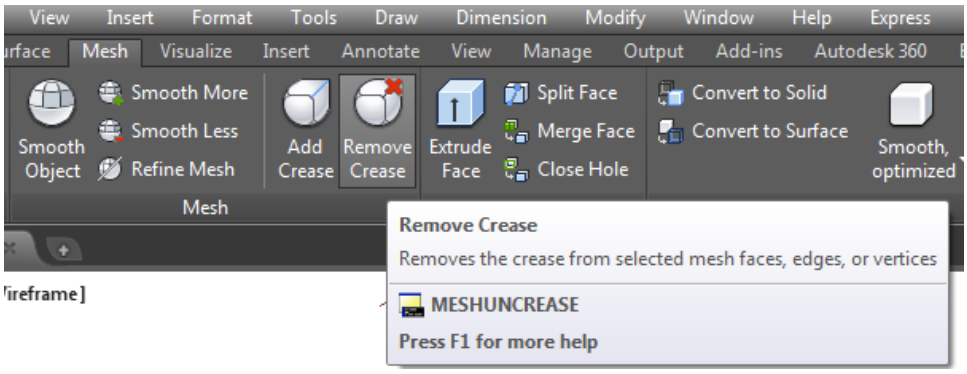
Mesh Uncrease

Remove the crease from selected mesh faces, edges, or vertices. Restore smoothness to an edge that has been creased. You can also remove a selected crease in the Crease area of Properties palette by changing the Type value to none.

Mesh Remove Crease Command for 3D Surface can be find out through following methods.

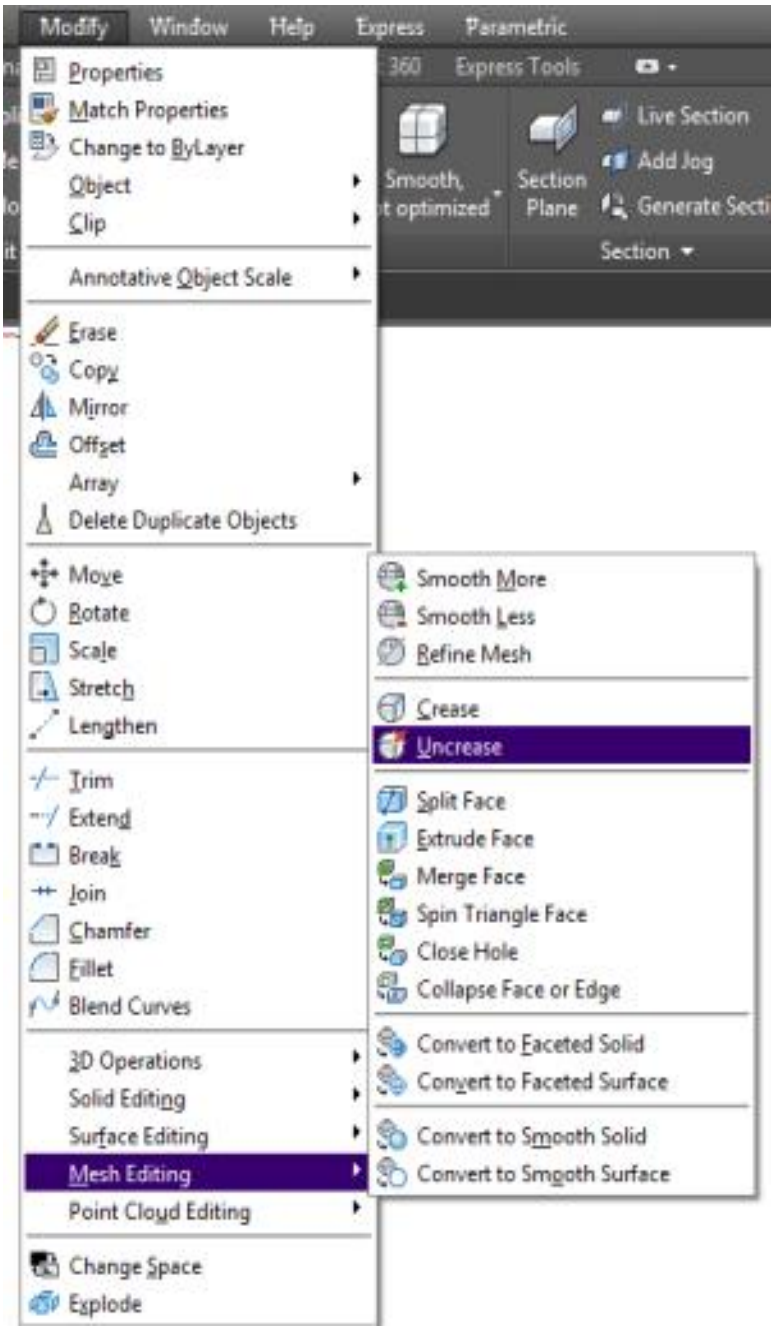
Through Ribbon

This figure describe the location of Mesh Remove Crease Command for 3D Surface in Mesh Tab



Through Main Menu

Main Menu →Modify →Mesh Editing →Uncrease



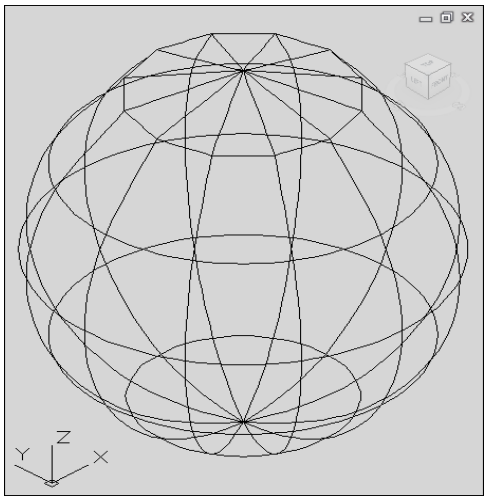
Through Short Cut Key

MESHUNCREASE → enter

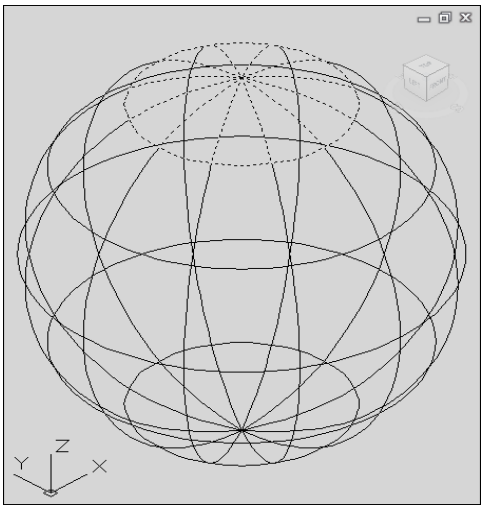
APPLICATION

Type MESHUNCREASE at the Command Prompt →Select Sub objects to convert → enter

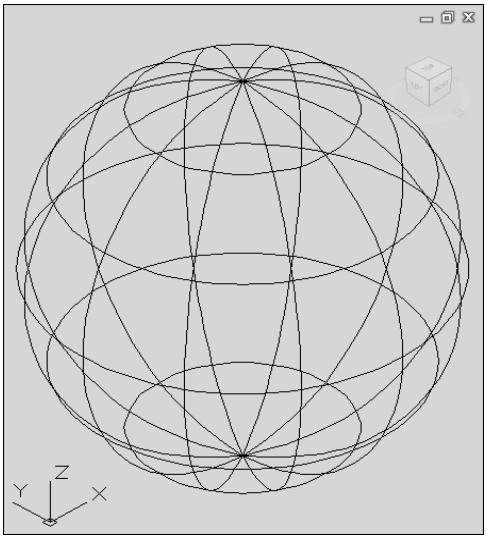
Before Uncrease Object



Selected Objects



After Uncrease Object



6. Editing Meshes

The following commands are used to the edit the mesh objects.

1. Extrude Face
2. Split Face
3. Merge Face
4. Close Hole
5. Collapse Face or Edge
6. Spin Traingle Face

Mesh Extrude

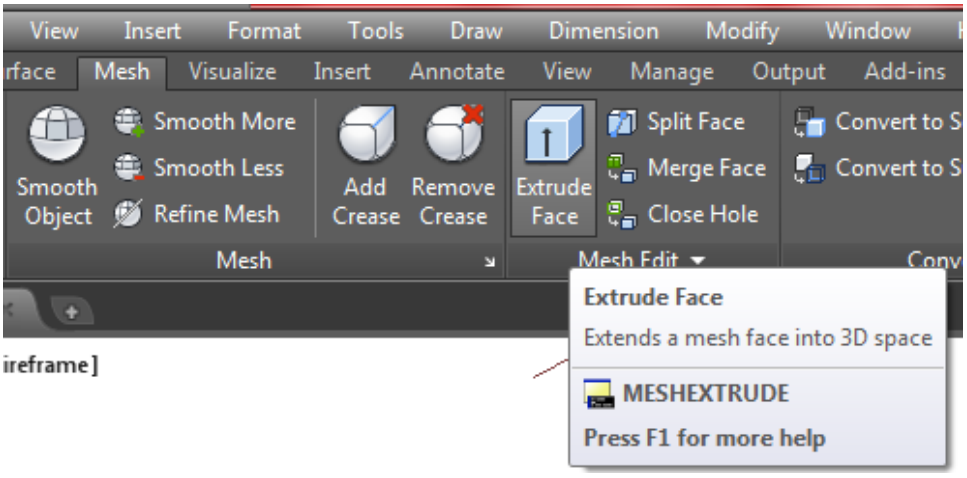
When you extrude, or extend, a mesh face, you can specify several options to determine the shape of the extrusion. You can also determine whether extruding multiple mesh faces results in joined or separate extrusions.

Extend a mesh face into 3D space. In most cases, if you extrude a closed object, the result is a new 3D solid. If you extrude an open object, the result is a surface. When you select a mesh face, the resulting extrusion distorts the face, but the extruded face continues to be part of the original object.

Mesh Extrude Face Command of 3D Surface can be find out through following methods.

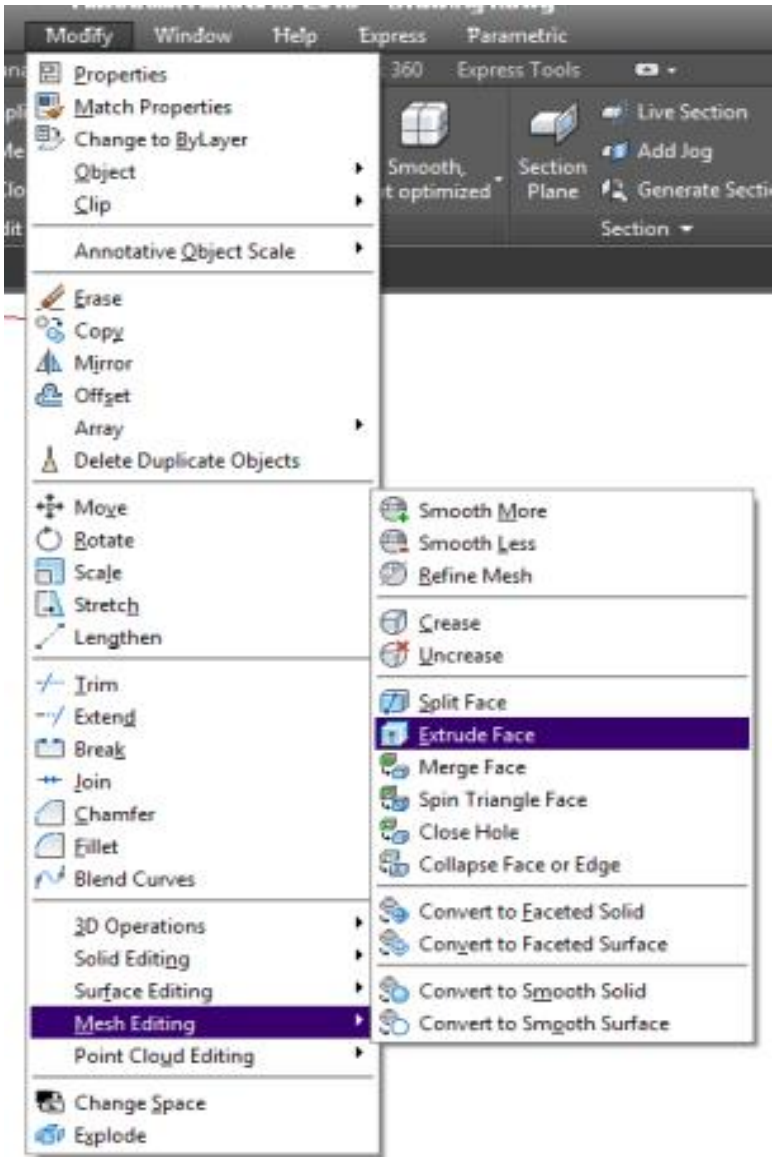
Through Ribbon

This figure describes the location of Mesh Extrude Face Command of 3D Surface in Mesh Tab.



Through Main Menu

Main Menu →Modify →Mesh Editing →Extrude Face



Through Short Cut Key

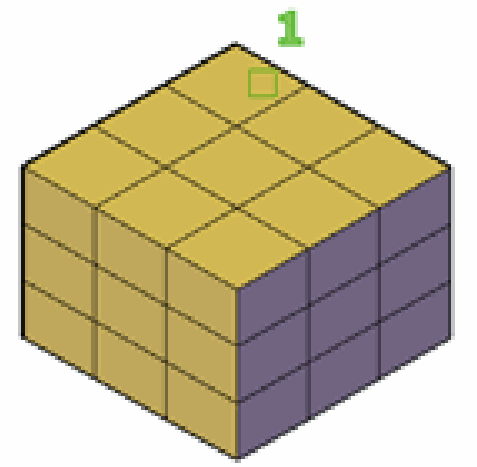
MESHEXTRUDE → enter

APPLICATION

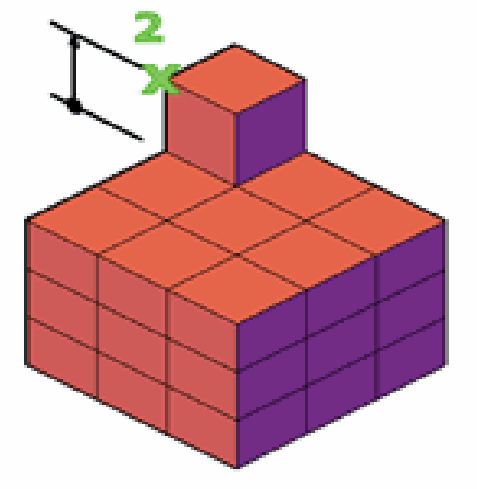
Type MESHEXTRUDE at the Command Prompt →Select Mesh Faces to Extrude → enter → Specify height of extrusion → enter Selected mesh face

Remember

You can select faces on solids and meshes by pressing and holding Ctrl, and then selecting the sub objects.



Extruded Mesh Face

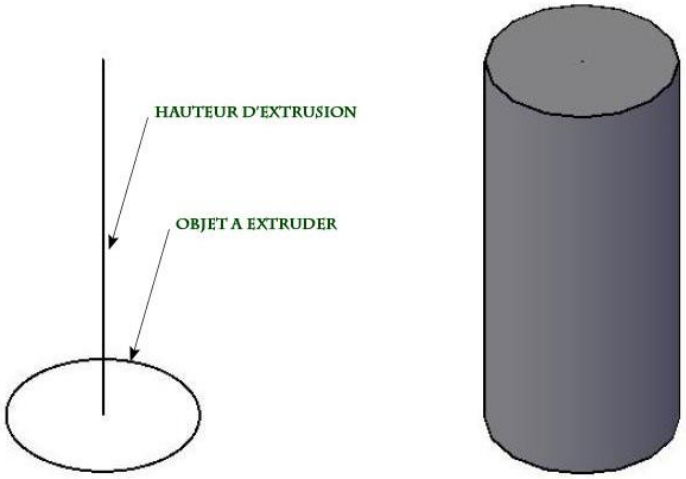


Mesh Extrude Options

a) Height of extrusion
b) Direction
c) Path
d) Taper angle

a) Height of extrusion

Extrudes objects along the positive *Z* axis of the object's coordinate system if you enter a positive value. The objects will be extruded along positive Z-axis. If you enter a negative value, the objects are extruded along the negative *Z* axis. Objects do not have to be parallel to the same plane. If all the objects are on a common plane, the objects are extruded in the direction of the normal of the plane.

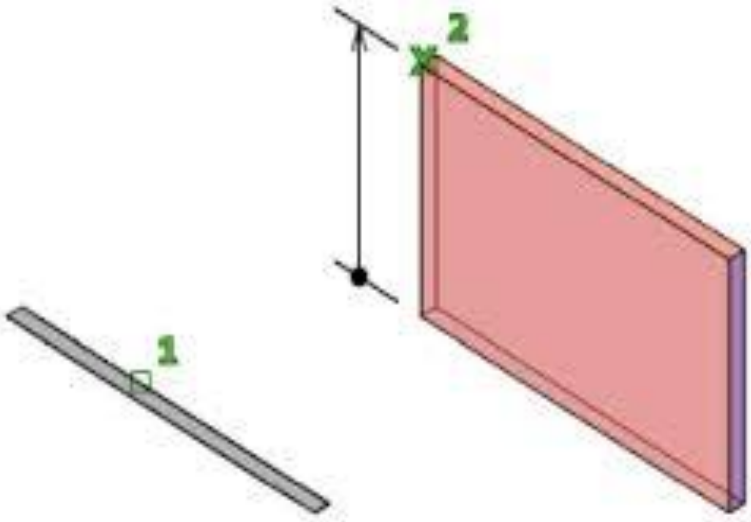


Do you know!

By default, planar objects are extruded in the direction of the object's normal.

b) Direction

Specifies the length and direction of the extrusion with two points you specify. (The direction cannot be parallel to the plane of the sweep curve created by the extrusion.)



Start point of direction. Specifies the first point in the direction vector.

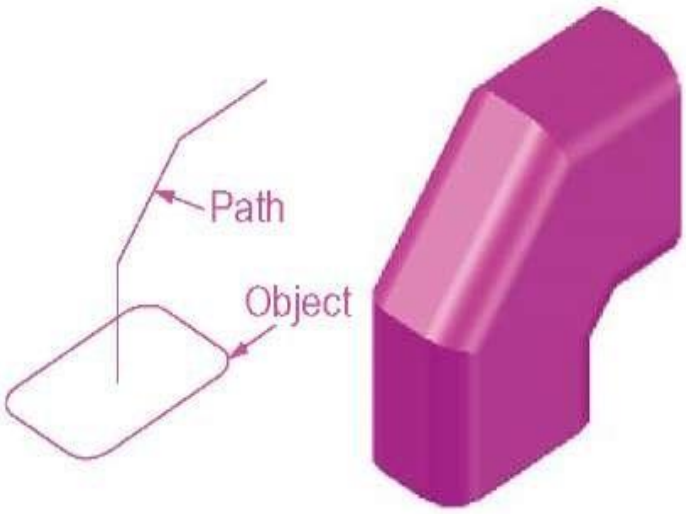
End point of direction. Specifies the second point in the direction vector.

c) Path

Specifies the extrusion path based on an object that you select. The path is moved to the centre of the profile. Then the profile of the selected object is extruded along the chosen path to create solids or surfaces.

Remember

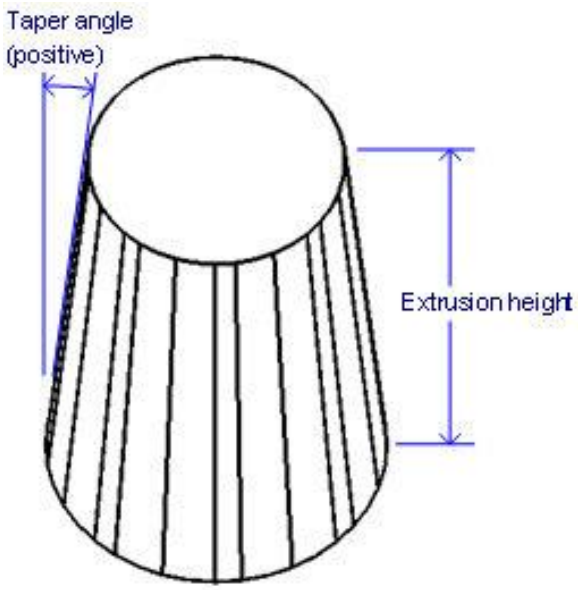
Press Ctrl+click to select faces and edges on 3D objects.



The path should not lie on the same plane as the object, nor should the path have areas of high curvature.

d) Taper angle

Set an angle of taper for an extrusion.



Positive angles taper in from the base object. Negative angles taper out. The default angle, 0, extrudes a 2D object perpendicular to its 2D plane. All selected objects and loops are tapered to the same value.

Specifying a large taper angle or a long extrusion height can cause the object or portions of the object to taper to a point before reaching the extrusion height. Individual loops of a region are always extruded to the same height. When an arc is part of a tapered extrusion, the angle of the arc remains constant, and the radius of the arc changes.

Angle of taper. Sets an angle between -90 and +90 degrees.

Specify two points. Sets the taper angle based on two points you specify in the drawing area. The taper angle applied to the extrusion is the distance between the two specified points.

Mesh Split

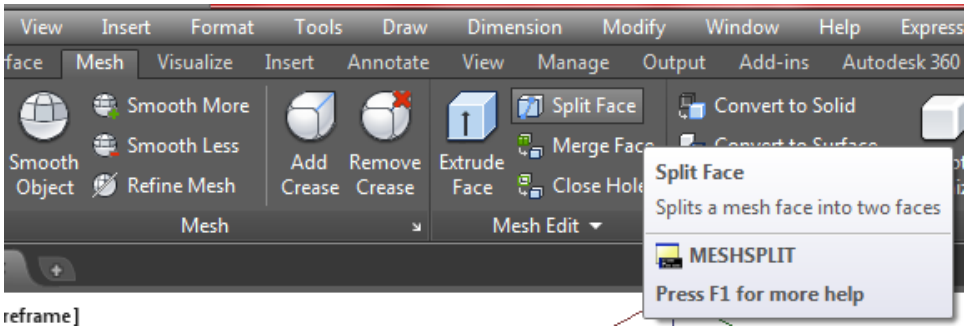
Split a face to add more definition to an area without having to refine it. Because you specify the start and endpoint of the split, this method provides greater control over the location of the split for more precision in the placement of the split, you can specify that the split starts or ends at a vertex. The Vertex option is useful for creating two triangular Faces from a rectangular face.

Split a mesh face into 2 faces. Split a face to add more definition to an area without having to refine it. Because you specify the start and endpoint of the split, this method provides greater control over the location of the split.

Mesh Split Face Command of 3D Surface can be find out through following methods.

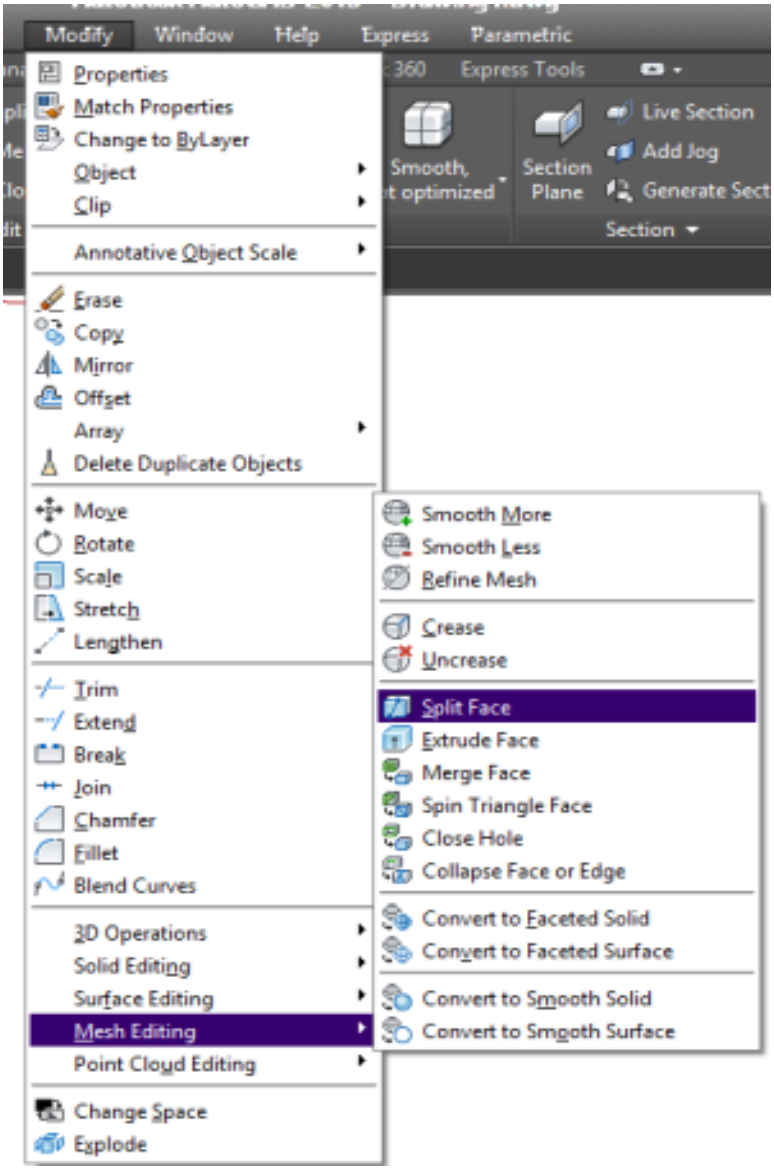
Through Ribbon

This figure describes the location of Mesh Split Face Command of 3D Surface in Mesh Tab.



Through Main Menu

Main Menu →Modify →Mesh Editing →Split Face



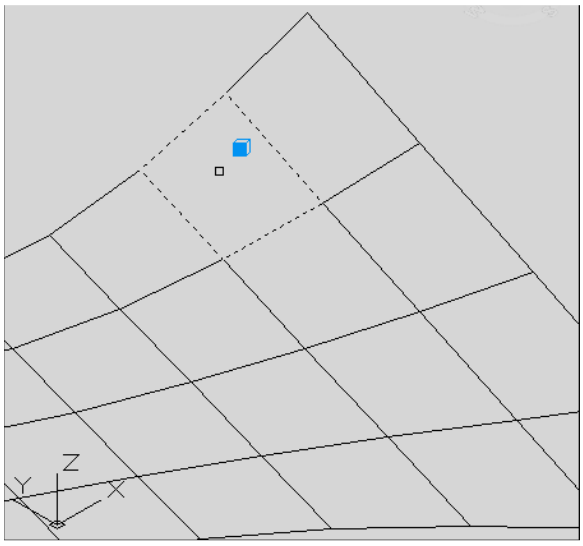
Through Short cut Key

MESHSPLIT → enter

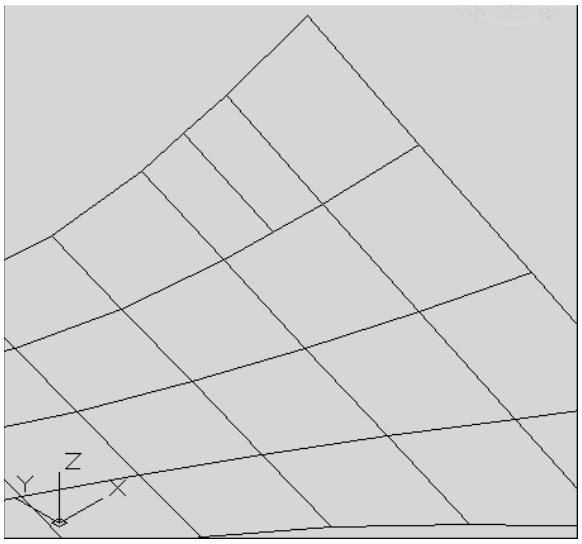
APPLICATION

Type MESHSPLIT at the Command Prompt →Select Mesh Faces to Split → Specify first split point on face edge → Specify second split point on face edge

Before Split



After Split



Merge Face

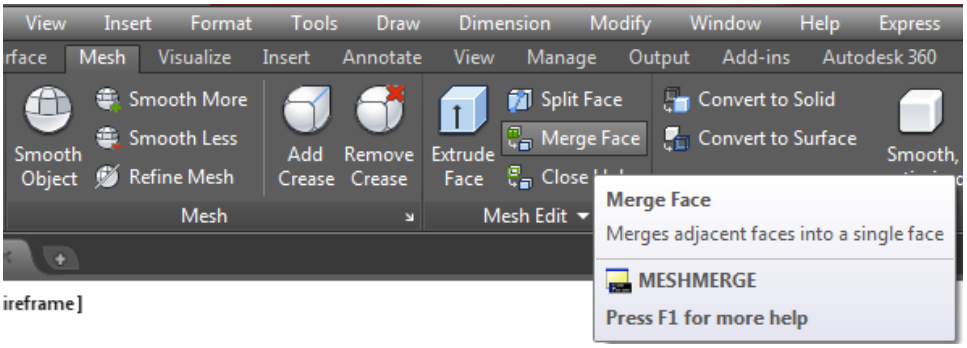
You can merge two or more adjacent mesh faces to form a single face. The merge operation is performed only on mesh faces that are adjacent. Other types of sub objects are removed from the selection set.

Merging mesh faces that wrap a corner can have unintended results when you try to edit the mesh or convert it to other types of solid objects.

Mesh Merge Face Command of 3D Surface can be find out through following methods.

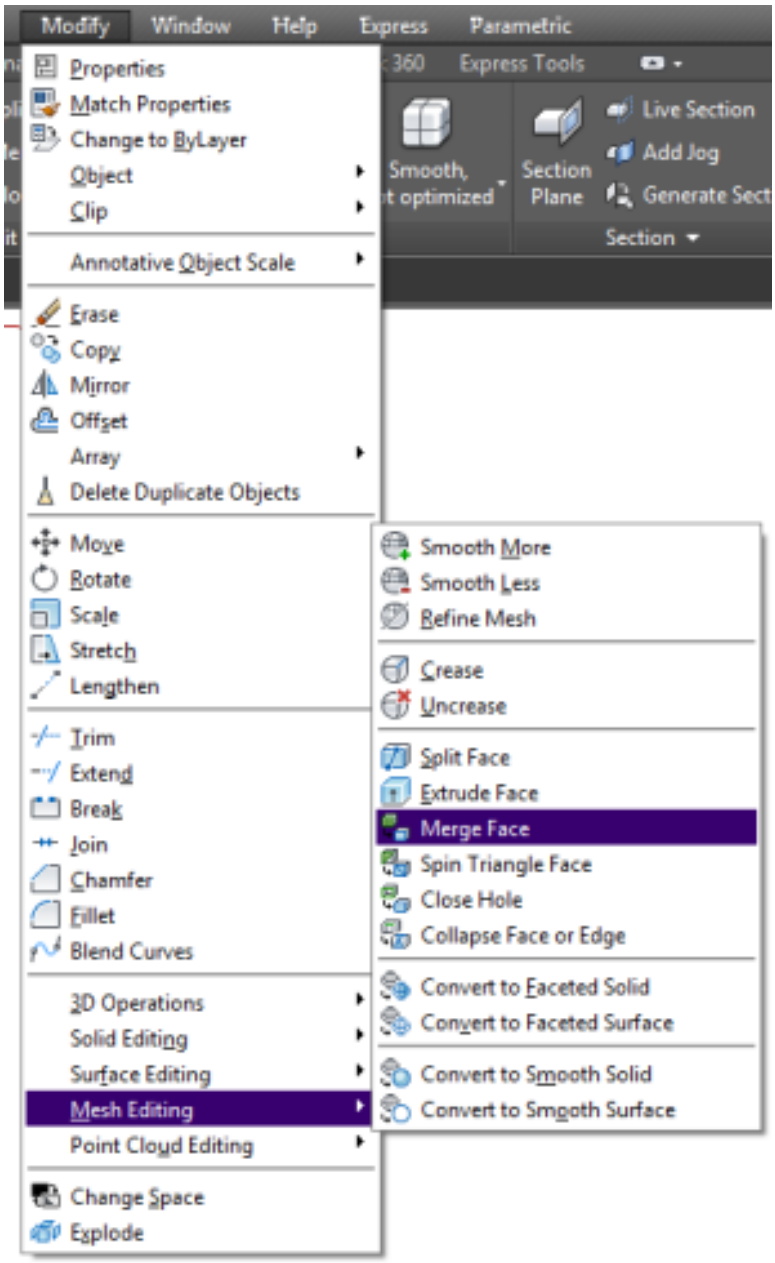
Through Ribbon

This figure describes the location of Mesh Merge Face command of 3D Surface in Mesh Tab



Through Main Menu

Main Menu →Modify →Mesh Editing→Marge Face



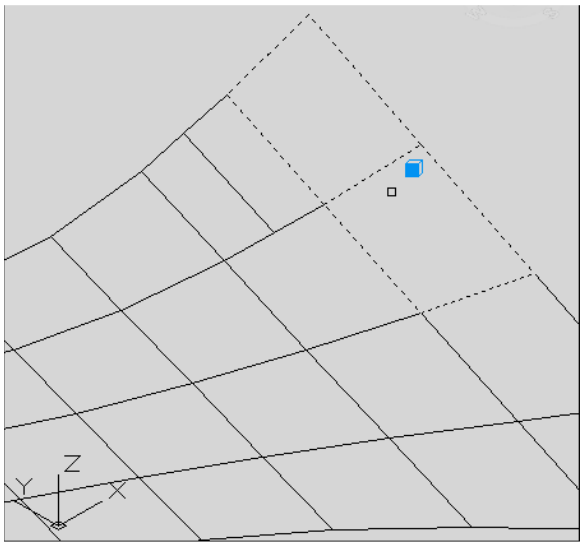
Through Short cut Key

MESHMERGE → enter

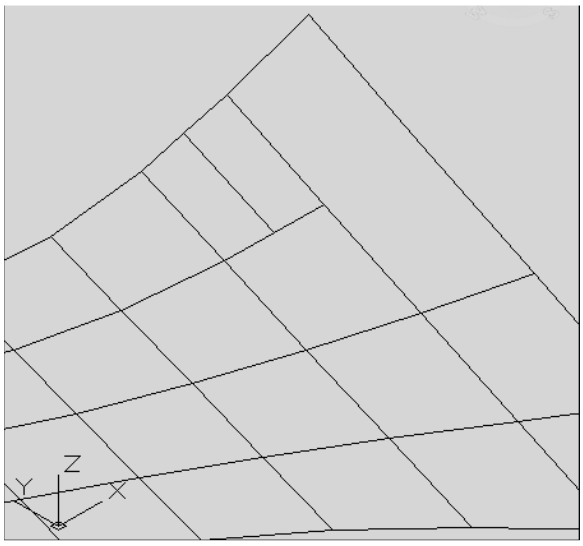
APPLICATION

Type MESHMERGE at the Command Prompt →Select Mesh Faces to Merge → Select first face to Marge → Select 2nd face to Marge→enter

Selected Mesh Faces



Resultant Merge



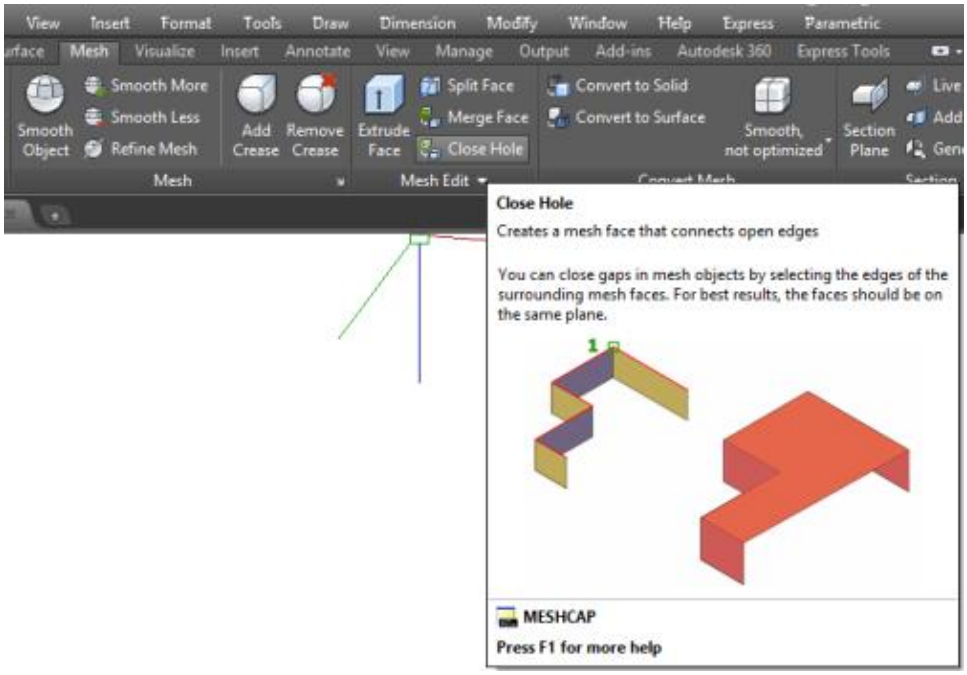
Mesh Cap (Close Hole)

Create a mesh face that connects open edges. You can close gaps in mesh objects by selecting the edges of the surrounding mesh faces. For best results, the faces should be on the same plane.

Mesh Close Hole Command of 3D Surface can be find out through following methods.

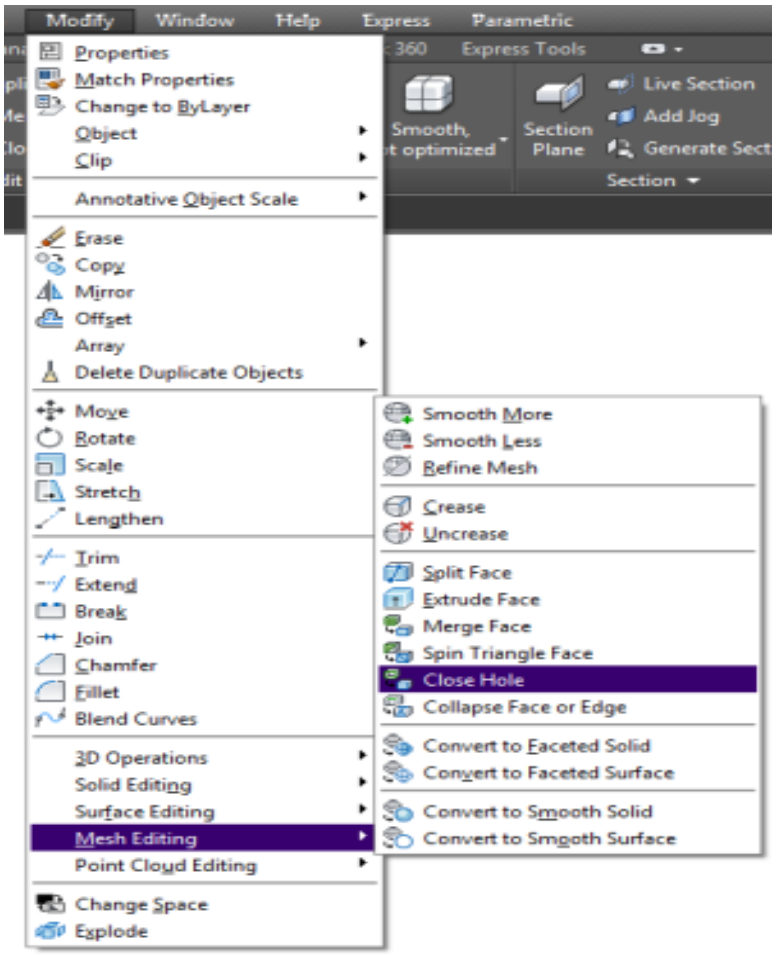
Through Ribbon

This figure describes the location of Mesh Close Hole Command of 3D Surface in Mesh Tab.



Through Main Menu

Main Menu →Modify →Mesh Editing →Close Hole



Through Short Cut Key

MESHCAP → enter

APPLICATION

Type MESHCAP at the Command Prompt → Select connecting mesh edges to create a new mesh face →enter

Mesh cap Options

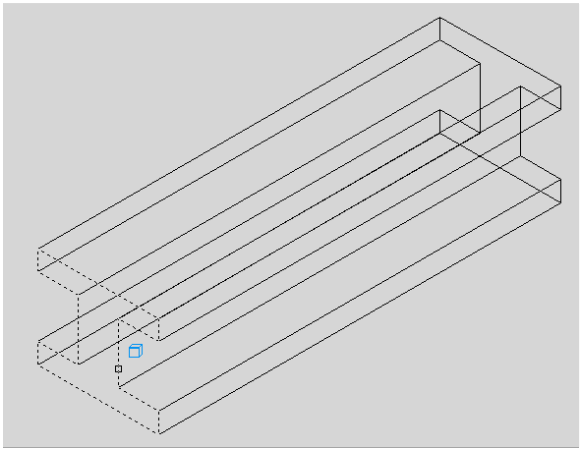
a) Edges

Select individual edges and adds them to the selection set.

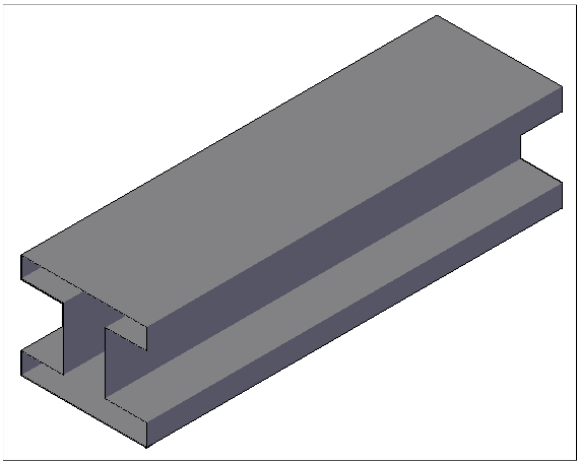
b) Chain

Select contiguous edges of mesh objects that are connected end-to-end

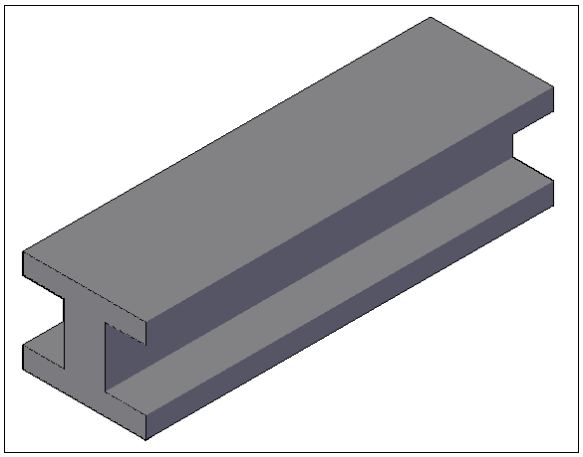
Selected Mesh Edges



Mesh Object before Close



Mesh Object after Close



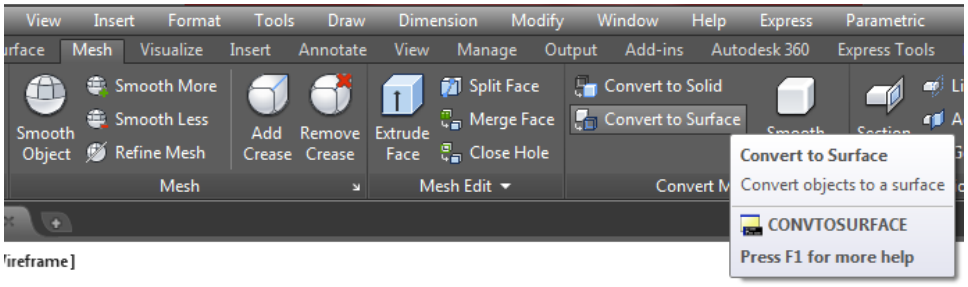
7. Converting Meshes

As you convert objects to surfaces, you can specify whether the resulting object is smooth or faceted. When you convert a mesh, the smoothness and number of faces of the resulting surface are controlled by the SMOOTHMESHCONVERT system variable. Whereas the previous example shows a conversion to a smooth, optimized surface, the following example shows a conversion to a faceted surface in which the faces are not merged, or optimized. With the CONVTOSURFACE command, you can convert the following objects into surfaces: 2D solids, 3D solids, Regions, open, zero-width polylines with thickness, lines with thickness, arcs with thickness, mesh objects, planar 3D faces.

Convert to Surface Command of 3D Surface can be find out through following methods.

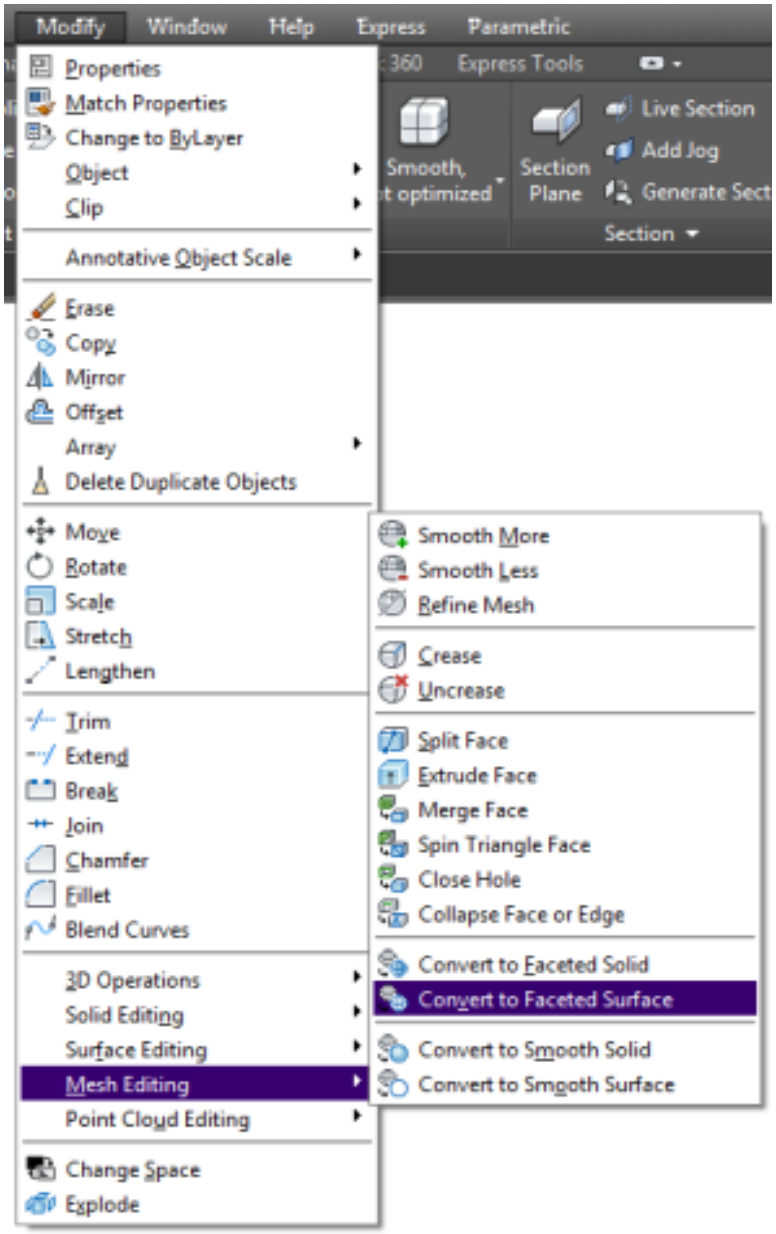
Through Ribbon

This figure describes the location of Convert to Surface command of 3D Surface in Mesh Tab



Through Main Menu

Main Menu →Modify →Mesh Editing →Convert to Surface



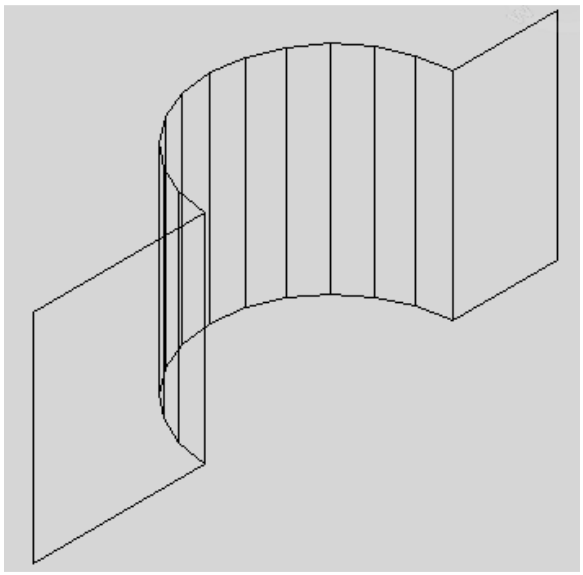
Through Short cut Key

CONVTOSURFACE → enter

APPLICATION

Type CONVTOSURFACE at the Command Prompt → Select Poly line with Thickness → enter

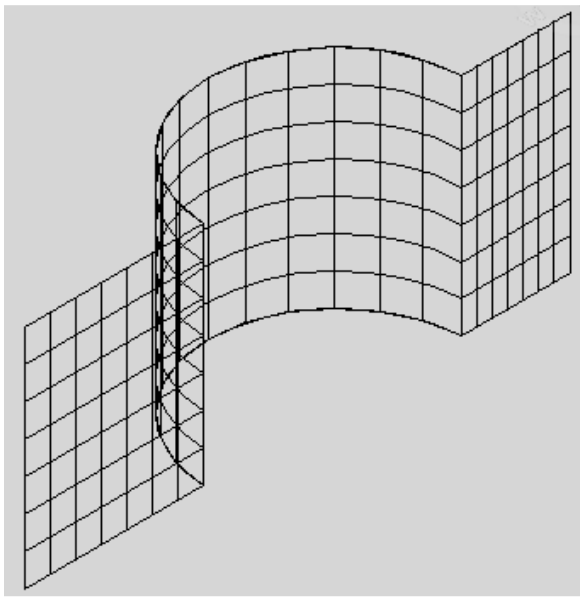
Poly Line with Thickness



Do you know!

You can create surfaces from 3D solids with curved faces, such as a cylinder, with the **EXPLODE** command.

Now Converted Mesh



Object That Can Be Converted to 3D Surfaces

a) 2D surface

b) Regions

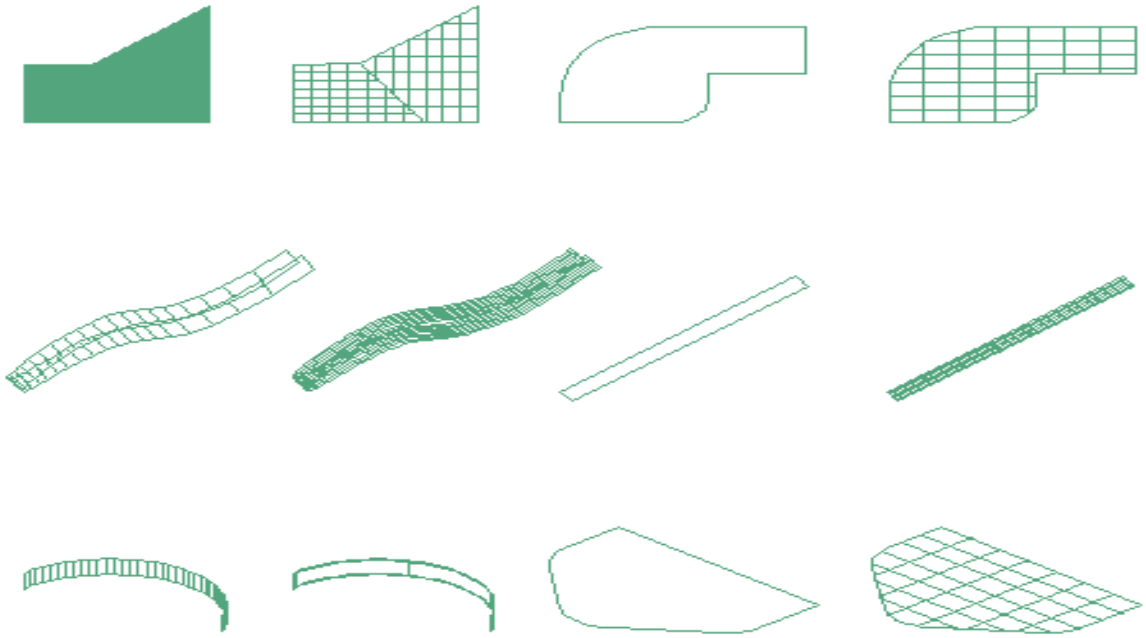
c) Open, zero-width polylines with thickness

d) Lines with thickness

e) Arcs with thickness

f) Mesh objects

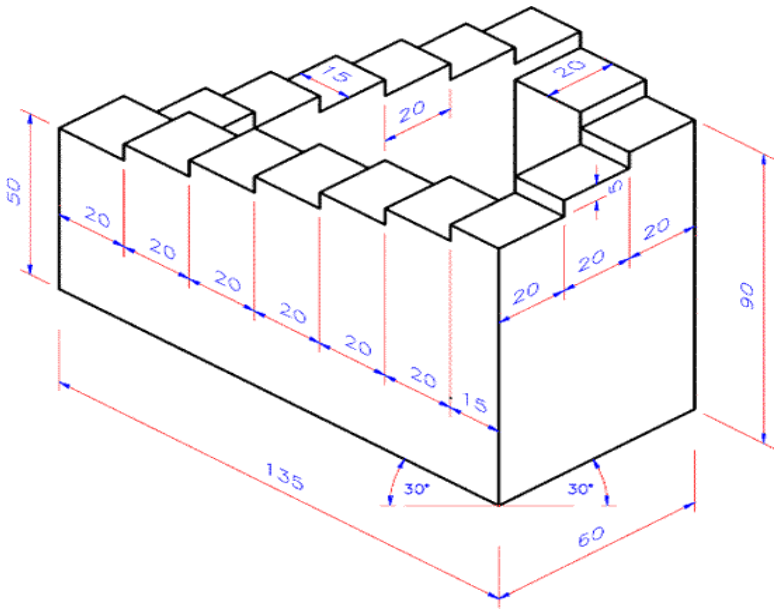
g) Planar 3D faces



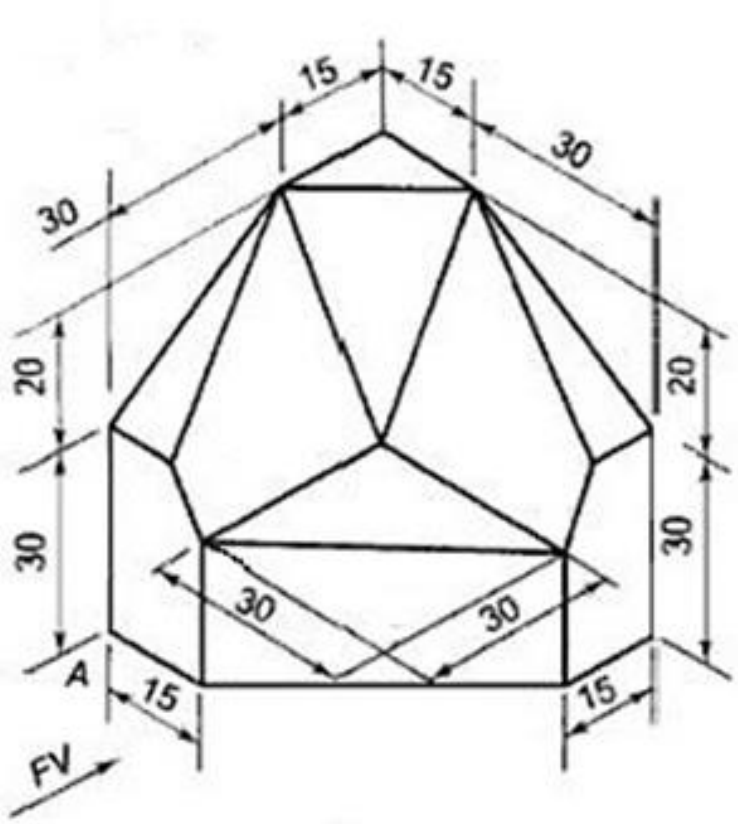
Learner's Activity

Use to meshes command to draw the flowing object

Assignment # 01



Assignment # 02



Assignment # 03

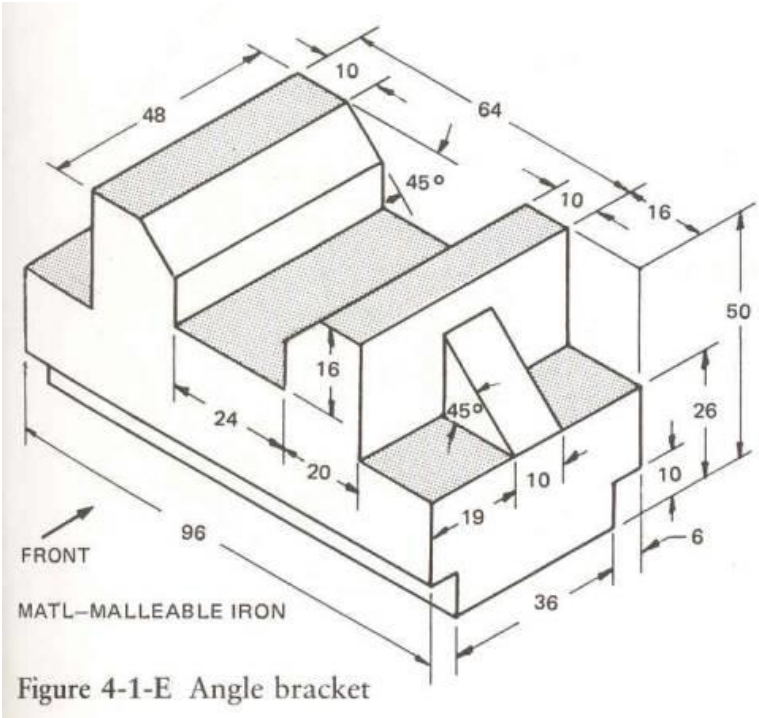


Figure 4-1-E Angle bracket

Learning Unit 2: Developing Complex Objects with 3D Surfaces

Overview

This unit describes how to develop complex objects having 3D surfaces. At the end of this unit, draw and edit complex surface. You will be able to revolve a line, Arc, 2D/3D Polyline surface around specified axis by using Surf tab's you could controlled the density of mesh. You could also tabulate surfaces, Ruled surface and adjust the edge surface .Moreover you could extrude surface in 2-axis. And you will also learn about loft Surface, their Setting, Sweep Surface, their alignment, Surface network, revolve Surface, Surface blend, Surface patches, Offset Surface. You will able to edit surfaces by applying different commands. Such as surface fillet, Surface Trim/un trim, Surface extrude and Surface Sculpt. You will be able to draw NURBS Surfaces, and Control NURBS vertex, finally you will be able to analysis Surface through following commands;

- Analysis Zebra helps you to analyze surface continuity.
- Analysis curvature ensure you that the model stays within a specified range
- Analysis draft helps you to evaluate the model whether it has adequate draft between a part and its mold as you display a color gradient into a surface
- Analysis options display option for Zebra, Curvature and draft Analysis

1. Revolves Surfaces

Select a line, arc, circle, or 2D or 3D polyline to sweep in a circular path around a selected axis. Regenerates a three-dimensional model with hidden lines

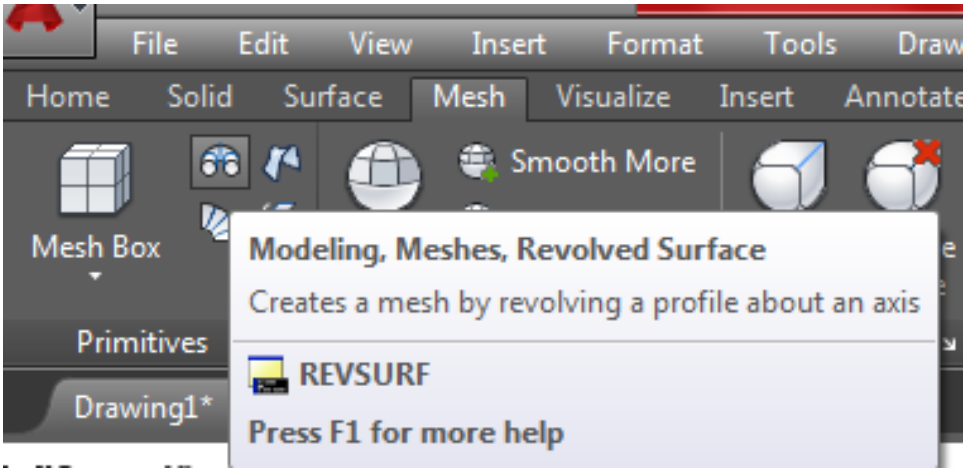
The MESHTYPE system variable sets which type of mesh are created. Mesh objects are created by default. Set the variable to 0 to create legacy polyface or polygon mesh.

The density of the generated mesh is controlled by the SURFTAB1 and SURFTAB2 system variables. SURFTAB1 specifies the number of tabulation lines that are drawn in the direction of revolution. If the path curve is a line, arc, circle, or spline-fit polyline, SURFTAB2 specifies the number of tabulation lines that are drawn to divide it into equal-sized intervals. If the path curve is a polyline that has not been spline fit, tabulation lines are drawn at the ends of straight segments, and each arc segment is divided into the number of intervals specified by SURFTAB2

Revolved Surface Command of 3D Surface can be find out through following ways

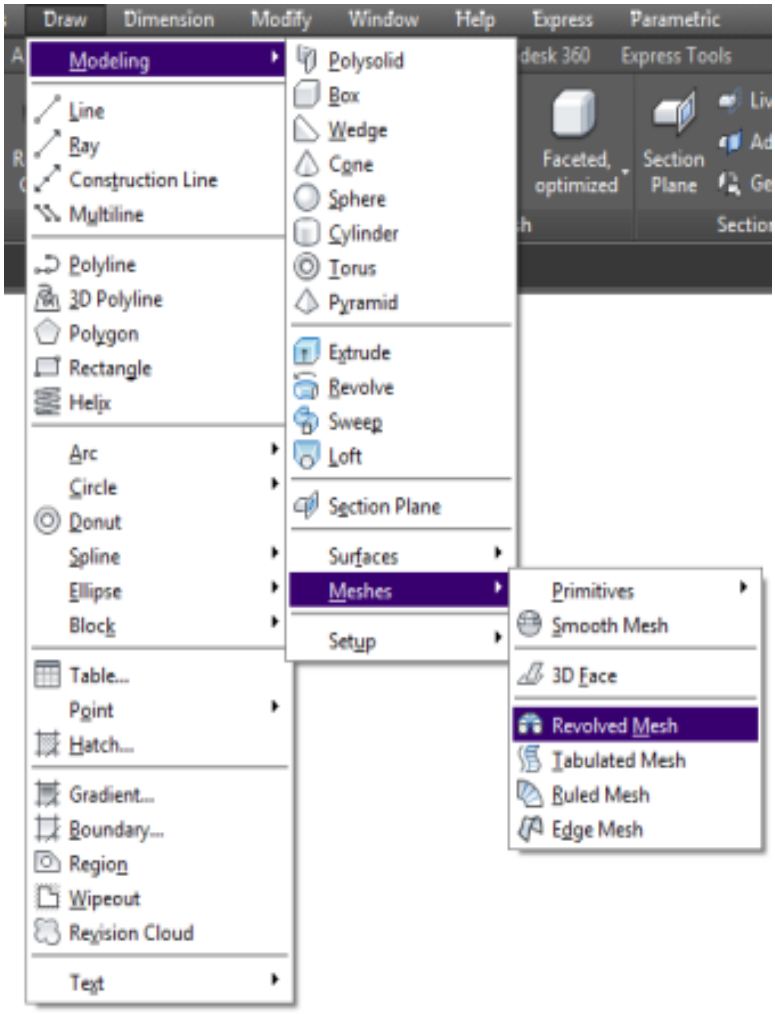
Through Ribbon

This figure describe the location of Revolved Surface Command of 3D Surface in Mesh Tab



Through Main Menu

Main Menu →Draw →Modeling →Meshes→Revolved Mesh



Through Short Cut Key

REVSURF → enter

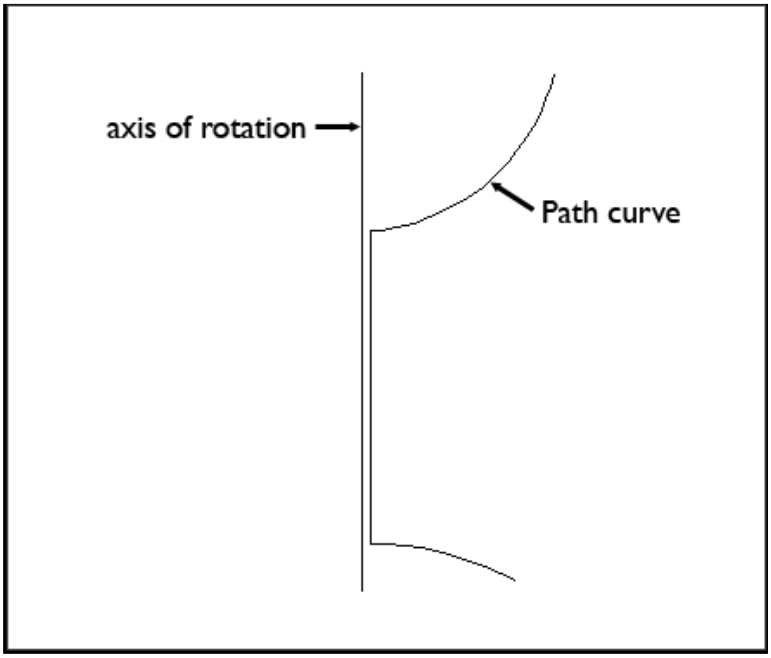
APPLICATION

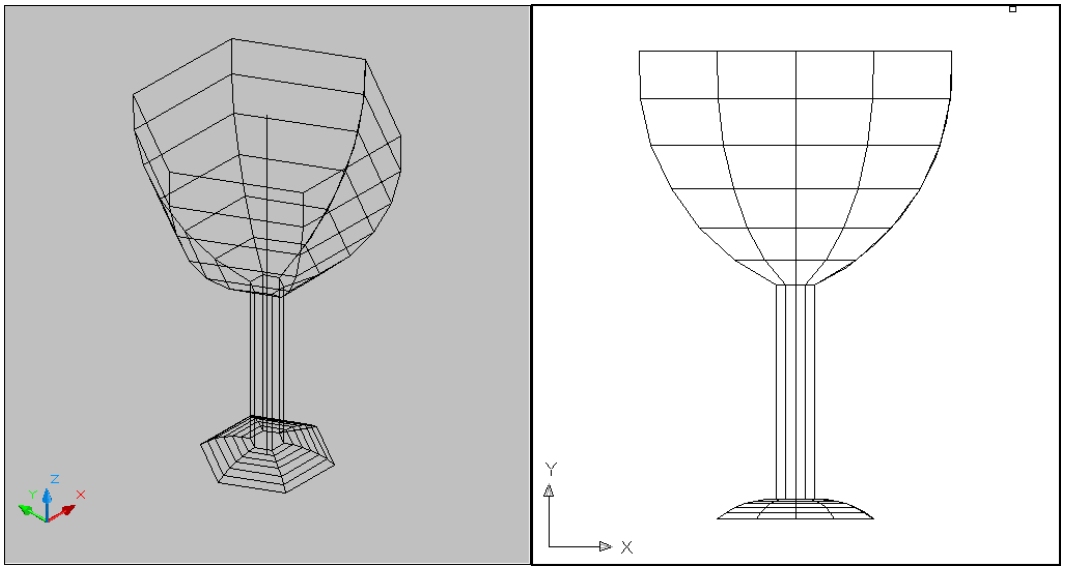
Type REVSURF at the Command Prompt → Select object to revolve→ Select object that defines the axis of revolution →Specify start angle → Specify included angle →enter

Do you know!

By Default value of

SURFTAB1=06
SURFTAB2=06





Surftab Variables

Set the number of tabulations for both directions to be generated for RULESURF and TABSURF. Also sets the mesh density in ROTATE3D the M direction for REVSURF and EDGESURF commands.

Method of change the Surftab1&2 values

APPLICATION

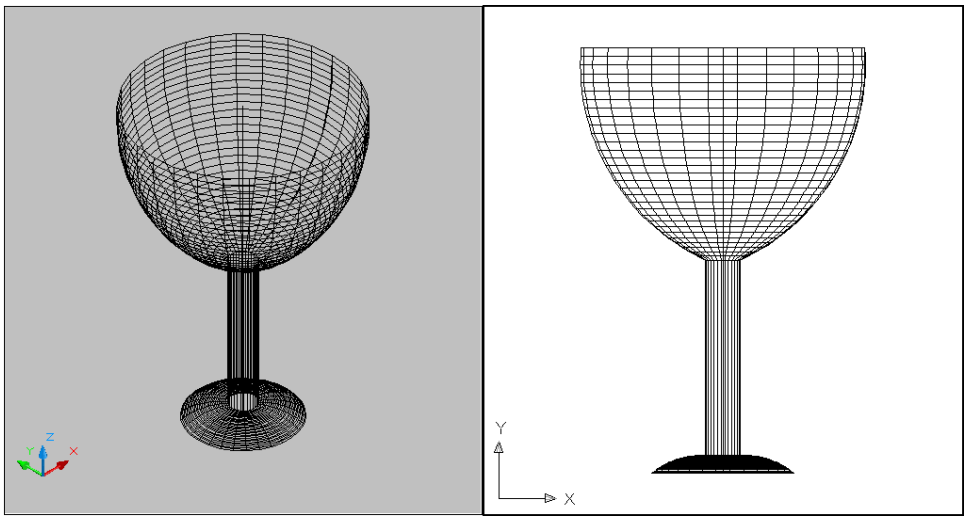
SURFTAB 1 Type **SURFTAB 1** at the Command Prompt → Enter new value for SURFTAB1 <6>:→ Put the new value of Surftab →enter

SURFTAB 2 Type **SURFTAB 2** at the Command Prompt → Enter new value for SURFTAB2 <6>:→ Put the new value of Surftab →enter

Reaction of Surftab Value on object

APPLICATION

Type REVSURF at the Command Prompt → Select object to revolve → Select object that defines the axis of revolution → Specify start angle → Specify included angle →enter



Note:
Change Value of
SURFTAB1=30
SURFTAB2=30

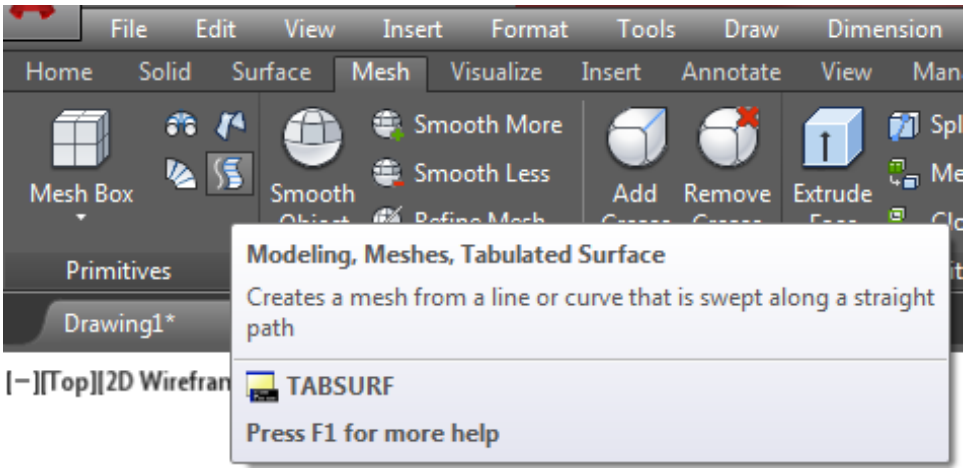
2. Tabulated Surfaces

Select a line, arc, circle, ellipse, or polyline to sweep in a straight path. Then select a line or polyline to determine the first and last points of a vector that indicates the direction and length of the polygon mesh.

Tabulated Surface Command of 3D Surface can be find out through following ways

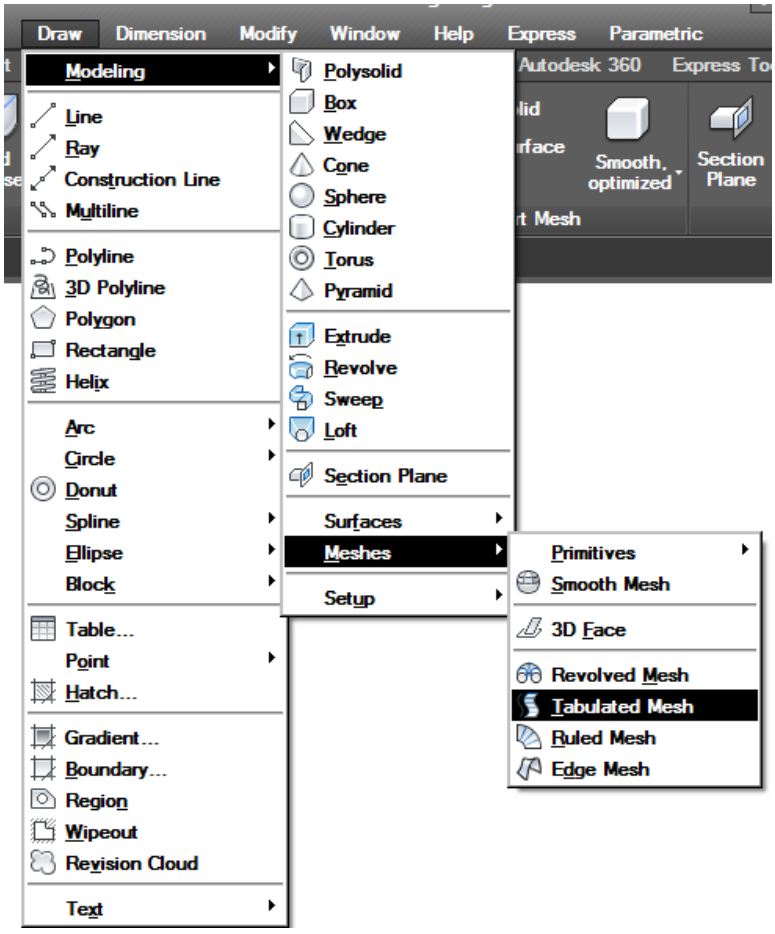
Through Ribbon

This figure describe the location of Tabulated Surface Command of 3D Surface in Mesh Tab



Through Main Menu

Main Menu →Draw →Modeling →Meshes→ Tabulated Mesh



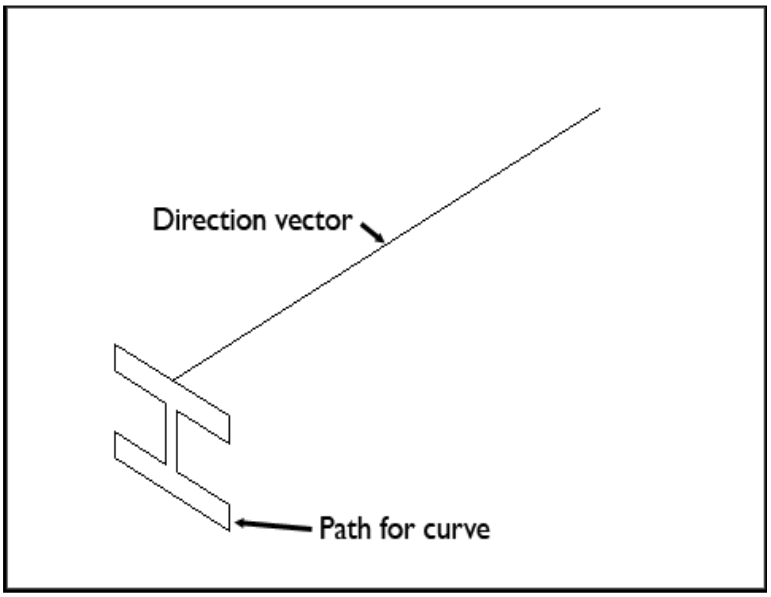
Through Short Cut Key

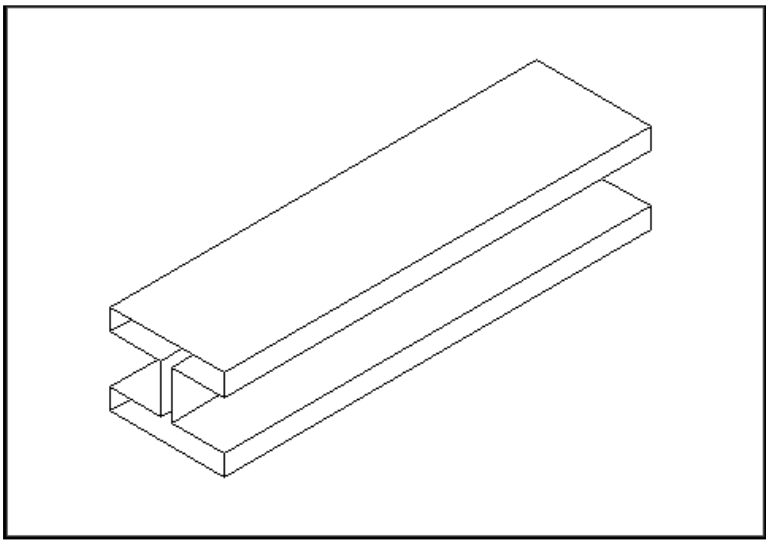
TABSURF → enter

APPLICATION

Type TABSURF at the Command Prompt → Select object for path curve → Select object for direction vector

Tabulated surface





Note:
Current wire frame density:
SURFTAB1=6

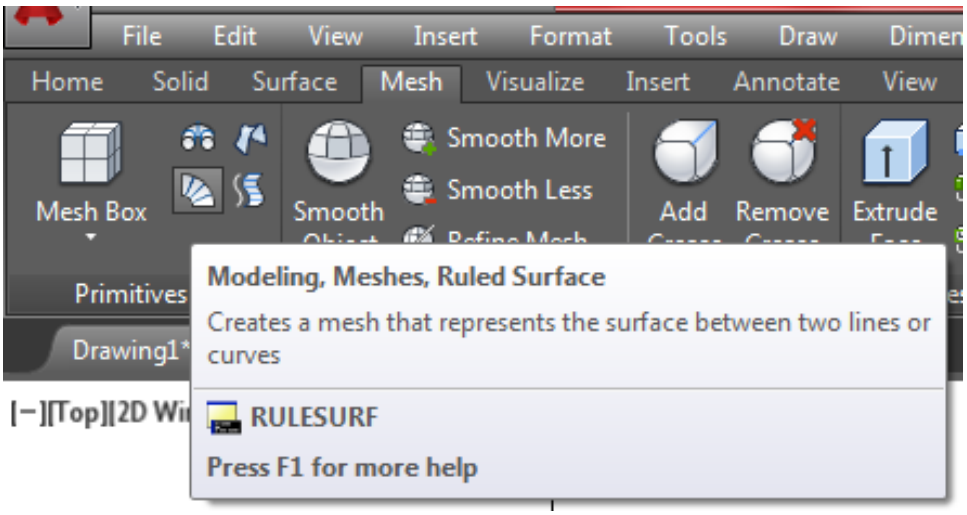
3. Ruled Surfaces

Select two edges that define the mesh. The edges can be lines, arcs, splines, circles, or polylines. If one of the edges is closed, then the other edge must also be closed. You can also use a point as one edge for either an open or a closed curve

Ruled Surface Command of 3D Surface can be find out through following methods.

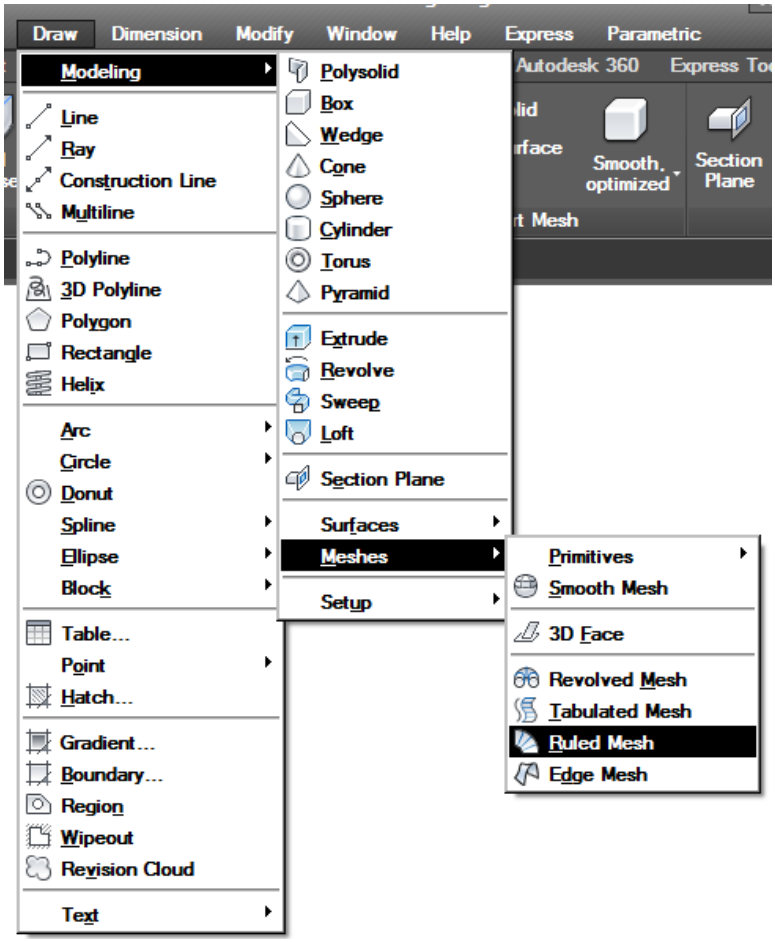
Through Ribbon

This figure describe the location of Ruled Surface Command of 3D Surface in Mesh Tab



Through Main Menu

Main Menu →Draw →Modeling →Meshes→Ruled Mesh



Through Short cut Key

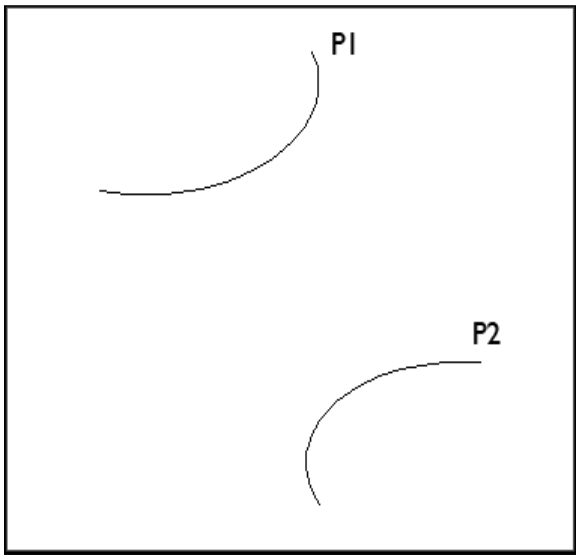
RULESURF → enter

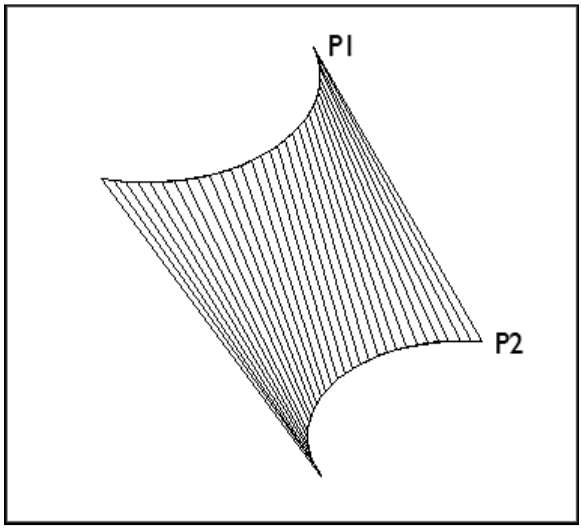
APPLICATION

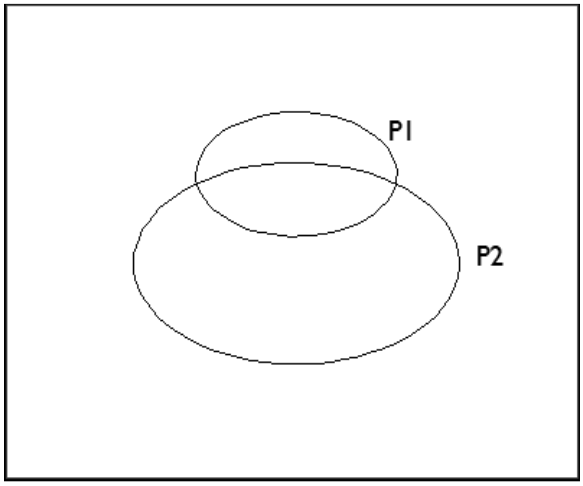
Type RULESURF at the Command Prompt → Select first defining curve: P1 → Select second defining curve: P2

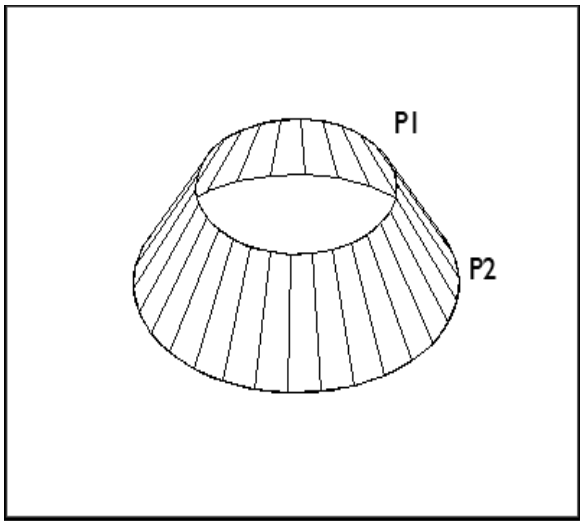
Rule surface

Note:
Current wire frame density:
SURFTAB1=6









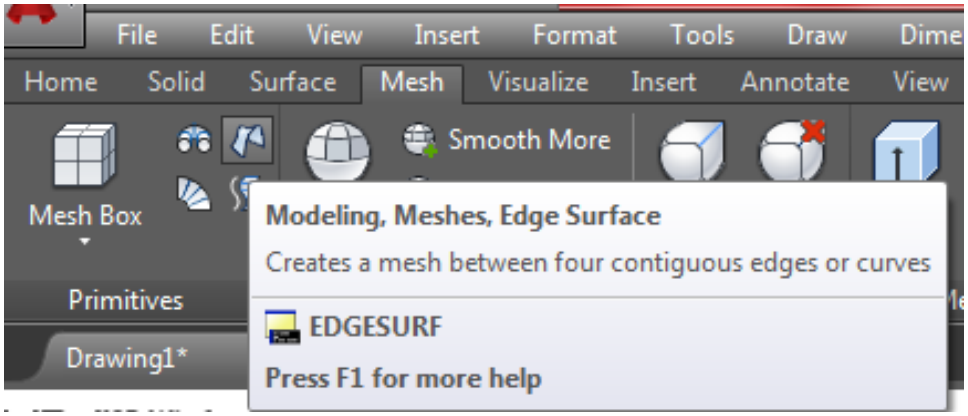
4. Edge Surfaces

Select four adjoining edges that define the mesh. The edges can be lines, arcs, splines, or open polylines. The edges must touch at their endpoints to form a single, closed loop

Edge Surface Command of 3D Surface can be find out through following ways

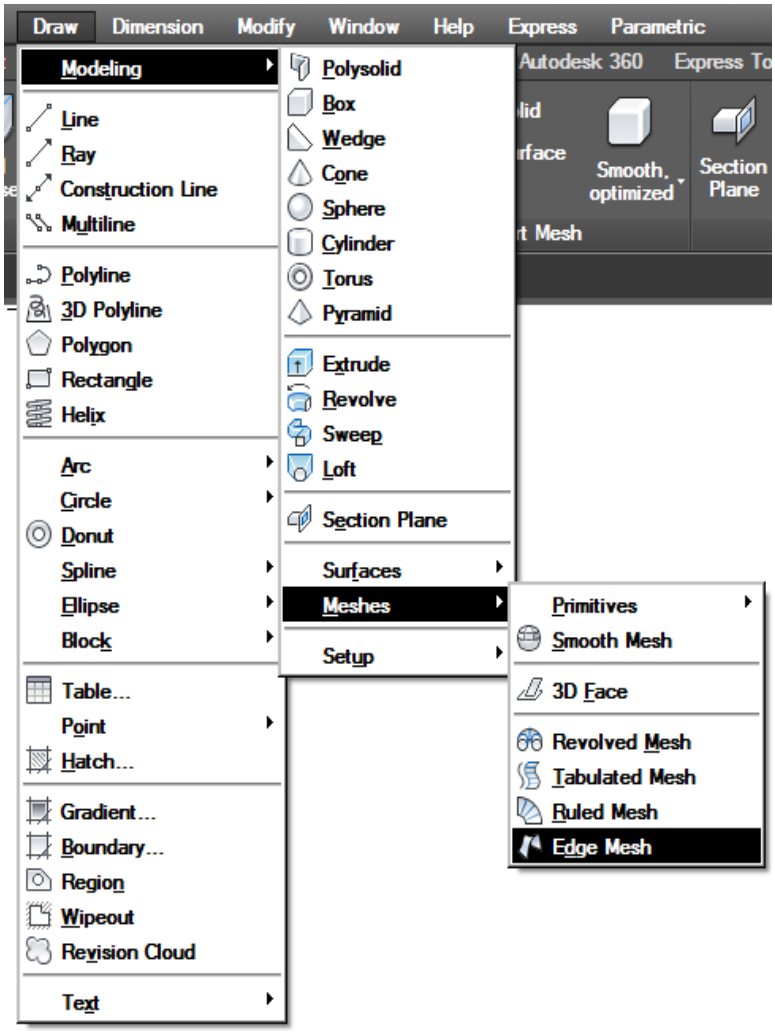
Through Ribbon

This figure describe the location of Edge Surface Command of 3D Surface in Mesh Tab



Through Main Menu

Main Menu →Draw →Modeling →Meshes→Edge Mesh



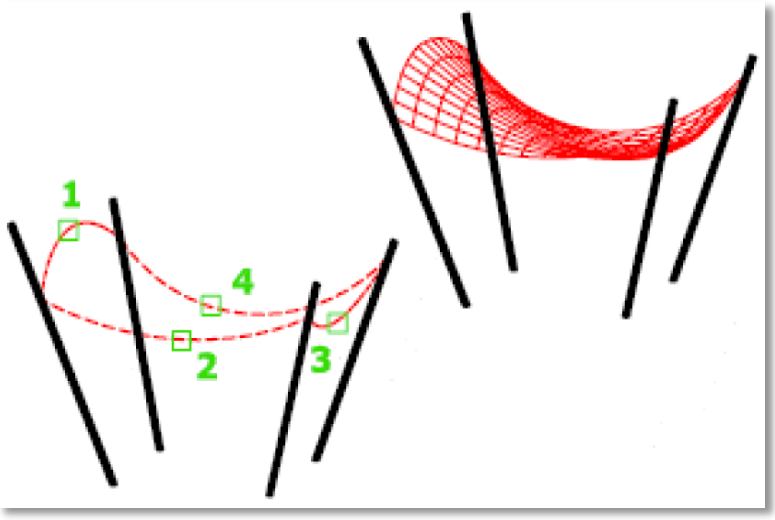
Through Short cut Key

EDGESURF → enter

APPLICATION

Type EDGESURF at the Command Prompt → Select object 1 for surface edge: P1 → Select object 2 for surface edge: P2 →Select object 3 for surface edge: P3→ Select object 4 for surface edge: P4

Edge surface



Note:
Current wire frame density:
SURFTAB1=20
SURFTAB2=10

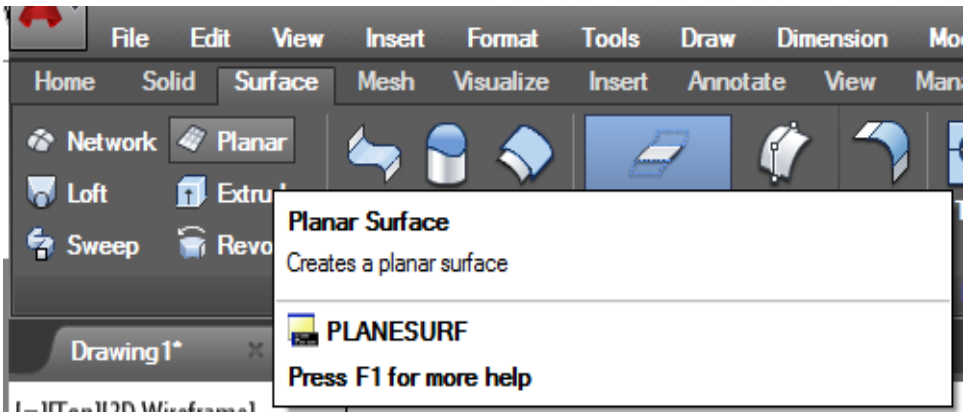
5. Plane Surfaces

You can create a plan surface by selecting closed objects or by specifying the opposite corners of a rectangular surface. Support pick-first selection and generates a plan surface out of a closed profile. When you specify the corners of the surface through the command, the surface is created parallel to the work plane. Create a planar surface by selecting a closed object or by creating a rectangular plane.

Planar Surface Command of 3D Surface can be find out through following methods.

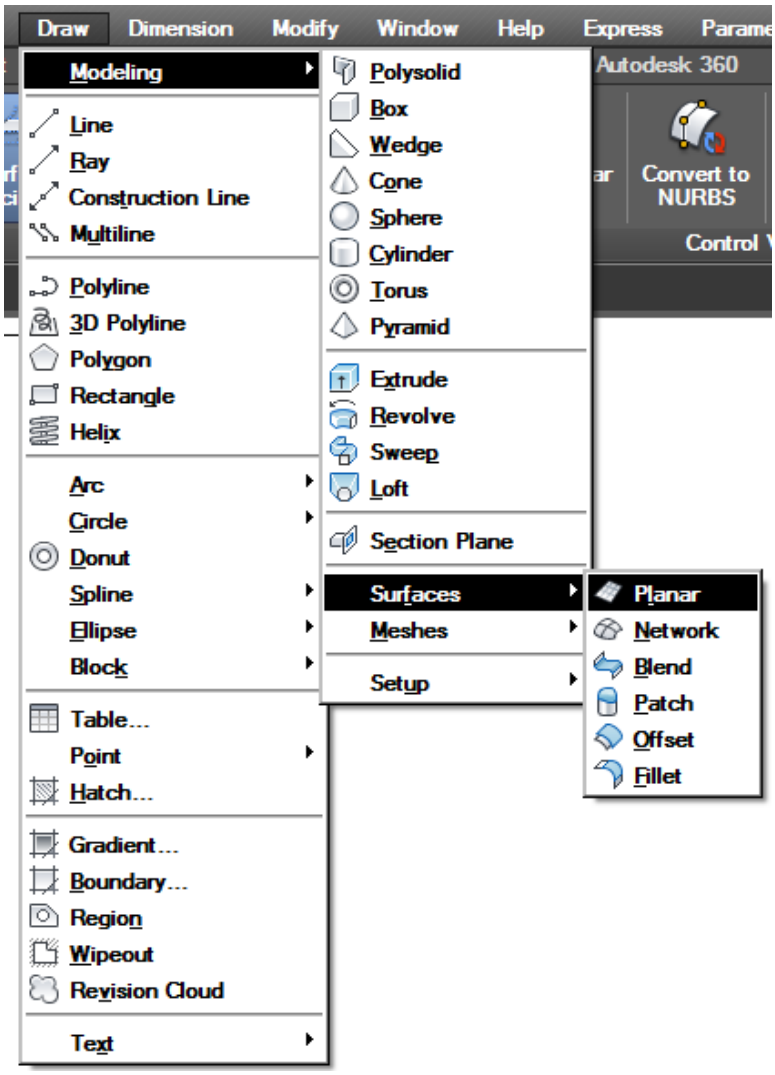
Through Ribbon

This figure describe the location of Planar Surface Command of 3D Surface in Surface Tab



Through Main Menu

Main Menu →Draw →Modeling →Surfaces→Planar



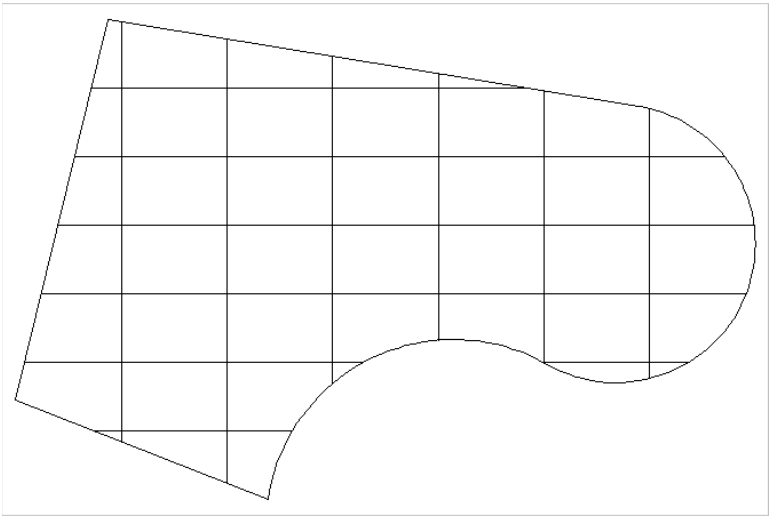
Through Short cut Key

PLANESURF → enter

APPLICATION

Type PLANESURF at the Command Prompt → Specify first corner or [Object] → O enter (for object) → Select object → enter

Planar surface



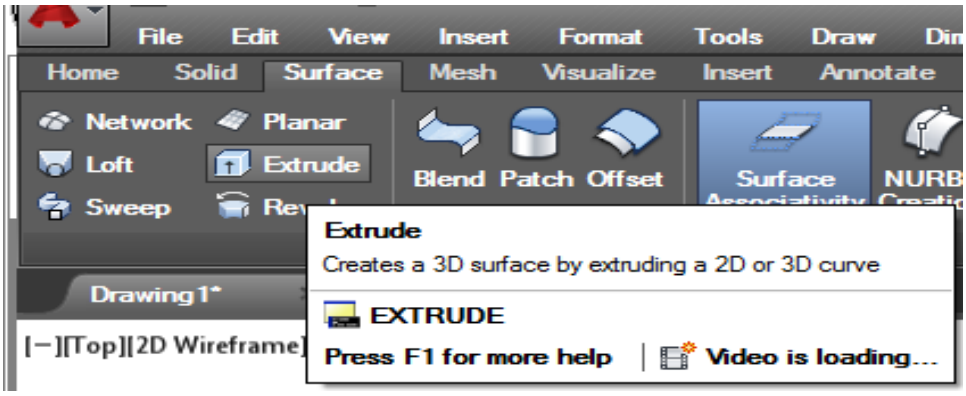
6. Extrude Surfaces

Extrusions can extend in the Z direction or be set to taper or follow a path. You can extrude an open or closed object to create 3D surface or solid.

Extrude Command of 3D Surface can be find out through following methods:

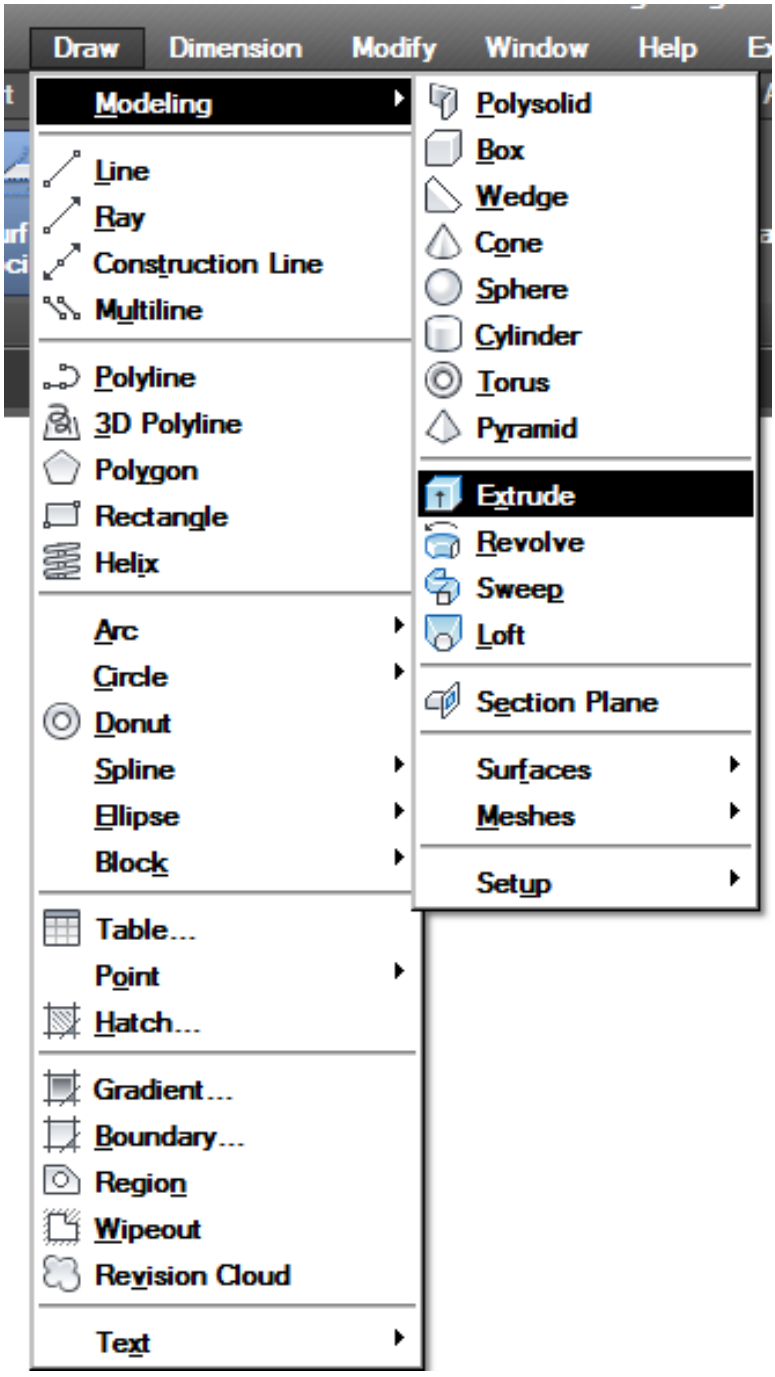
Through Ribbon

This figure describe the location of Extrude Command of 3D Surface in Surface Tab



Through Main Menu

Main Menu →Draw →Modeling →Extrude

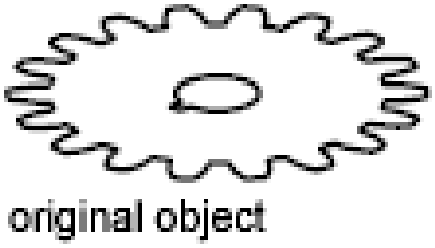


Through Short cut Key

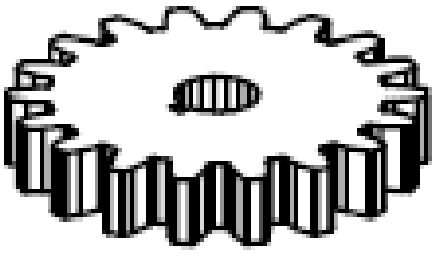
EXTRUDE → enter

APPLICATION

Type EXTRUDE at the Command Prompt → Select objects to extrude or [MOde] → MO enter (for Creation mode) →[SOlid/SUrface] Solid>→ SU enter (for Surface)→ Select objects to extrude or [MOde]: → Select objects→Specify height of extrusion or [Direction/Path/Taper angle/Expression] →put the value of height→Enter



original object



extruded object

Remember

You can select faces on solids and meshes by pressing and holding Ctrl, and then selecting the sub objects.

Extrude Options

a) Height of extrusion
b) Direction
c) Path
d) Taper angle

Detail about these options has already been discussed

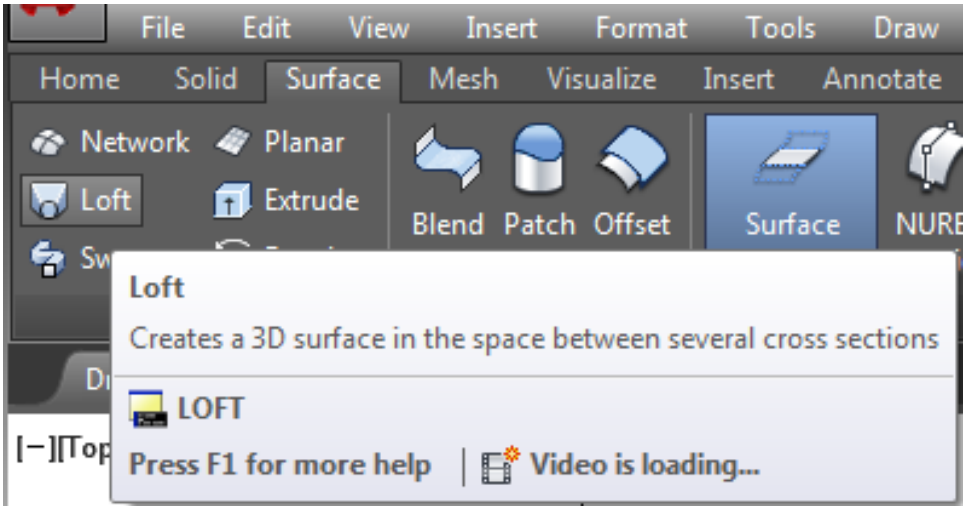
7. Loft Surfaces

Create 3D solid or surface by specifying a series of cross sections. The cross sections define the shape of the resulting solid or surface. You must specify at least two cross sections. Loft profiles can be open or closed, planar or non-planar, and can also be edge sub objects. Use the mode option to select whether to create a surface or a solid.

Loft Command of 3D Surface can be find out through following methods:

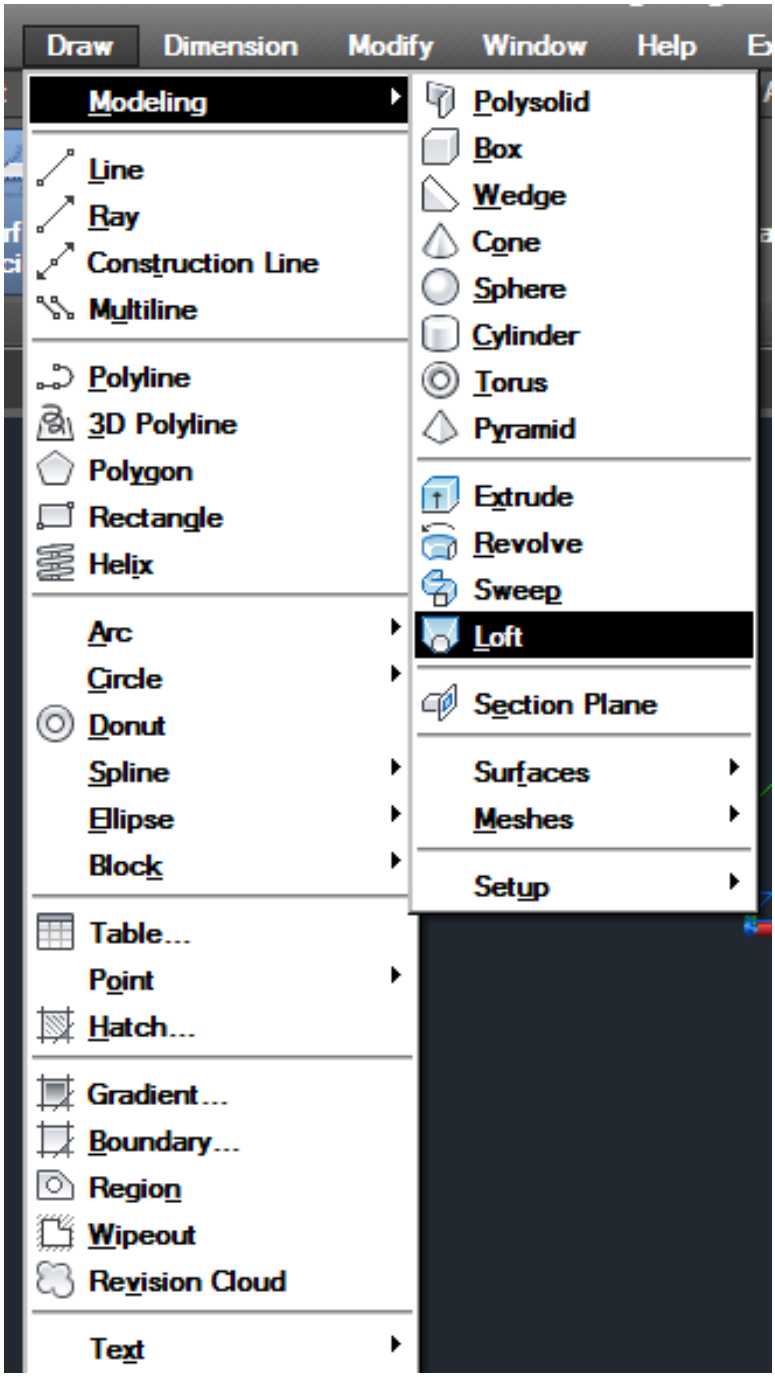
Through Ribbon

This figure describe the location of Loft Command of 3D Surface in Surface Tab



Through Main Menu

Main Menu →Draw →Modeling →Loft



Through Short Cut Key

LOFT → enter

APPLICATION

Type LOFT at the Command Prompt →

Select cross sections in lofting order or

[POint/Join edges/MOde]: select object →

Select cross sections in lofting order or

[POint/Join edges/MOde]: select object →

Select cross sections in lofting order or

[POint/Join edges/MOde]: select object →

Select cross sections in lofting order or

[POint/Join multiple edges/MOde]:

3 cross sections selected → Enter → option

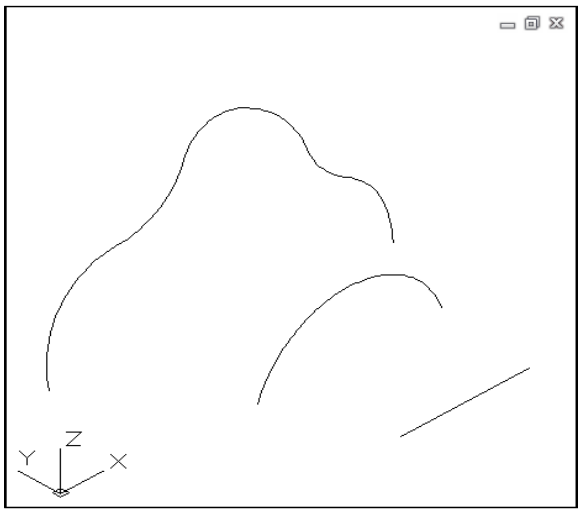
[Guides/Path/Cross sections only/Settings] →

Cross sections only→enter

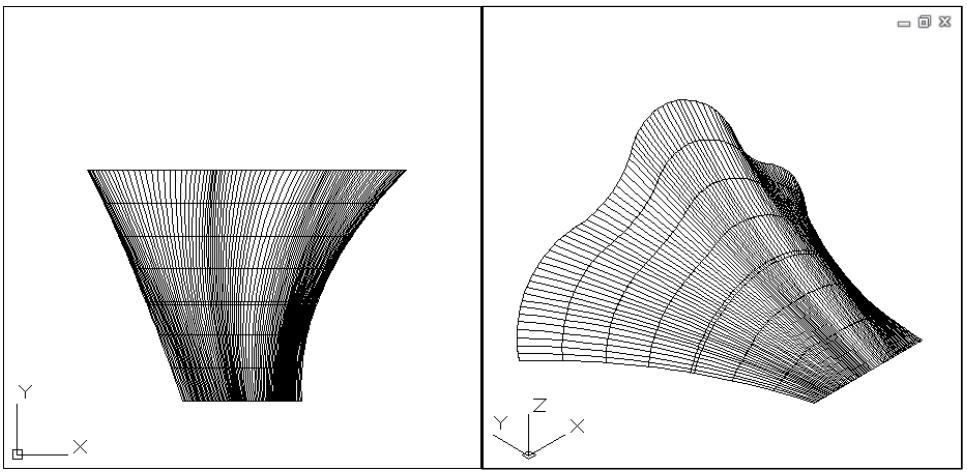
Objects before

Remember

Current wire frame density: ISOLINES=4,



Surfaces using Loft

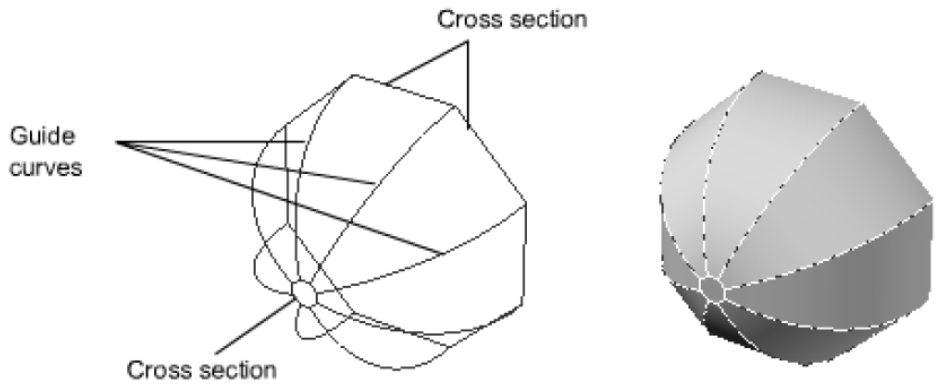


Loft Command Options

i. **Guides**
ii. **Path**
iii. **Cross sections only**
iv. **Settings**

i. Guides

Specifies guide curves that control the shape of the lofted solid or surface. You can use guide curves to control how points are matched up on corresponding cross sections to prevent undesired results, such as wrinkles in the resulting solid or surface.



Cross sections with guide curves

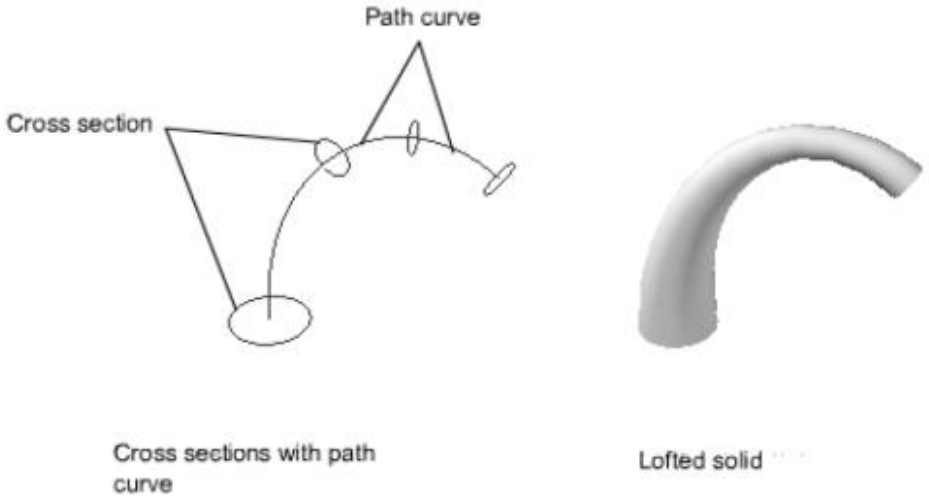
Lofted solid

- Intersects each cross section
- Starts on the first cross section
- Ends on the last cross section

Select any number of guide curves for the lofted surface or solid and press Enter.

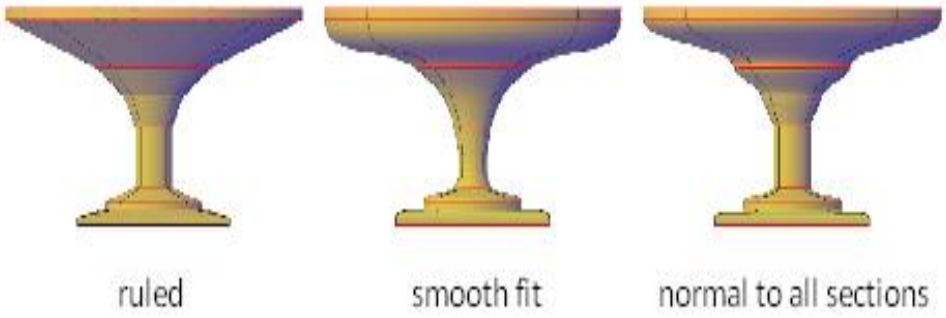
ii. Path

Specify a path for the loft operation to obtain more control over the shape of the lofted object. For best results, start the path curve on the plane of the first cross section and end it on the plane of the last cross section.



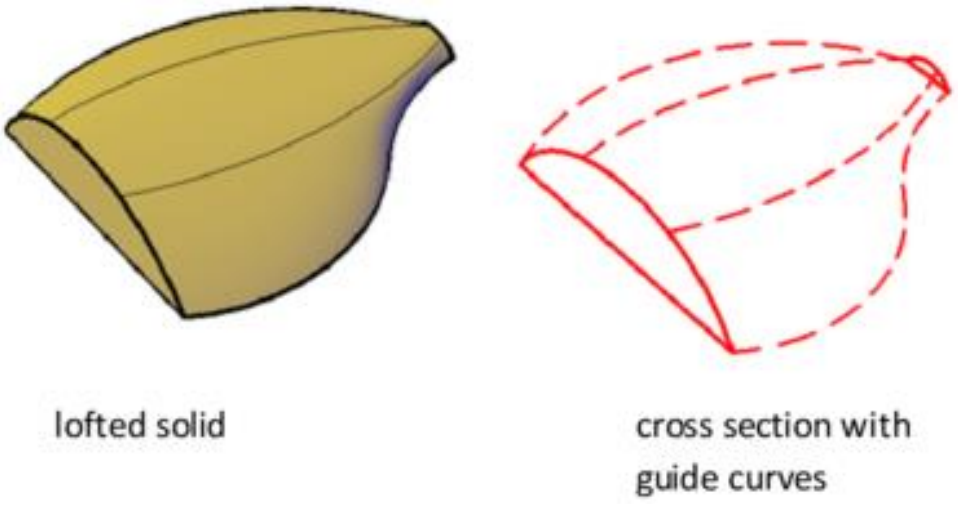
iii. Cross-section

Select a series of cross-section profiles to define the shape of the new 3D object. As you create a lofted object; you can adjust its shape by specifying how the profile passes through the cross sections (for example, a sharp or smooth curve). You can also modify the settings later in the Properties Inspector.



Guide curves.

Specifies guide curves to match points on corresponding cross sections. This method prevents undesired results, such as wrinkles in the resulting 3D object.



Each guide curve must meet the following criteria

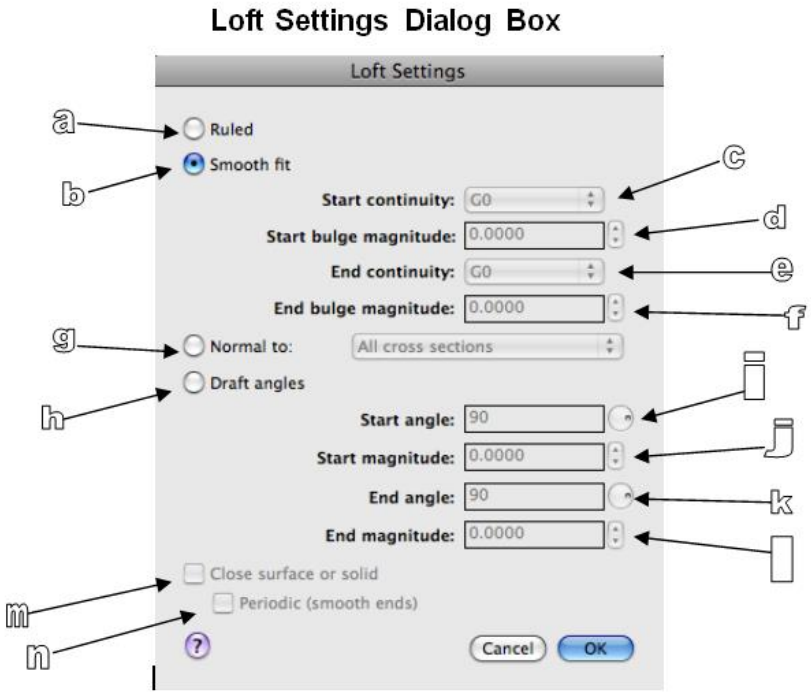
- Intersects each cross section
- Starts on the first cross section
- Ends on the last cross section

iv. Settings

Displays the **Loft Settings dialog box**.

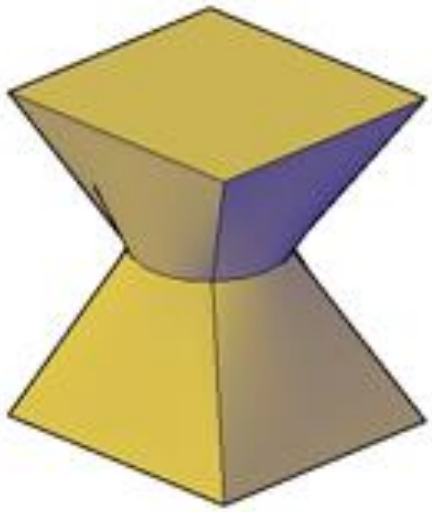
Control the contour of a lofted surface at its cross sections.

Also allows you to close the surface or solid.



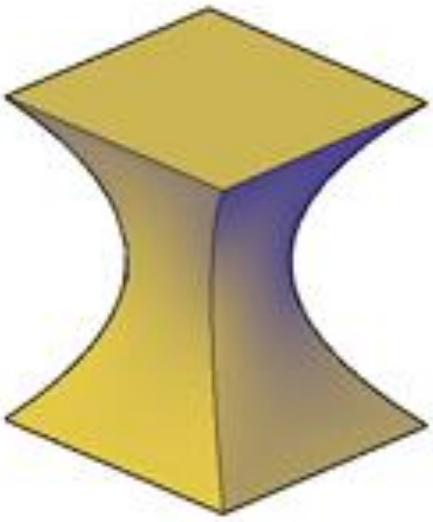
a) Ruled

Specify that the solid or surface is ruled (straight) between the cross sections and has sharp edges at the cross sections.



b) Smooth Fit

Specify that a smooth solid or surface is drawn between the cross sections and has sharp edges at the start and end cross sections.



c) Start Continuity

Set the tangency and curvature of the first cross section.

d) Start Bulge Magnitude

Set the size of the curve of the first cross section.

e) End Continuity

Set the tangency and curvature of the last cross section.

f) End Bulge Magnitude

Set the size of the curve of the last cross section.

g) Normal To

Controls the surface normal of the solid or surface where it passes through the cross sections

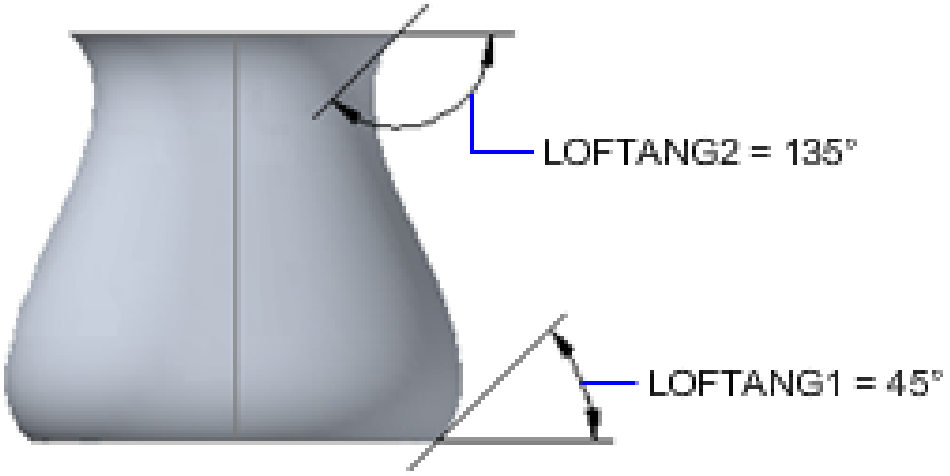
a. **Start Cross Section**
   Specify that the surface normal is normal to the start cross section.
b. **End Cross Section**
   Specify that the surface normal is normal to the end cross section.
c. **Start and End Cross Sections**
   Specifies that the surface normal is normal to both the start and end cross sections.
d. **All Cross Sections**
   Specify that the surface normal is normal to all cross sections.

h) Draft Angles

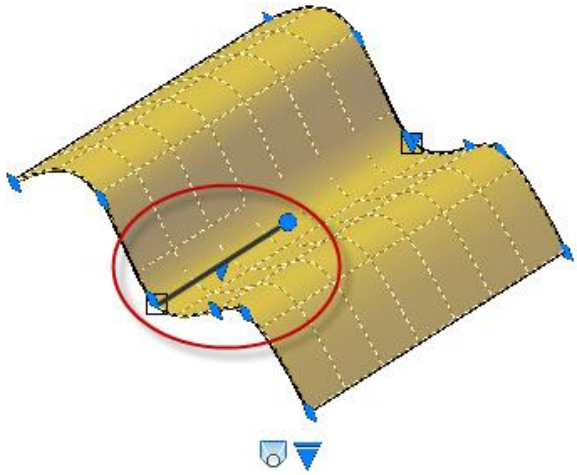
Control the draft angle and magnitude of the first and last cross sections of the lofted solid or surface. The draft angle is the beginning direction of the surface. 0 is defined as outward from the plane of the curve.

Draft angles set to 0 | Draft angles set to 90 | Draft angles set to 180

The following illustration shows the effect of using a different draft angle for the firstand last cross sections of a lofted solid. The first cross section is assigned a draft angle of 45 degrees, while the last cross section is assigned a draft angle of 135 degrees.



You can also use the draft angle handle to adjust the draft angle (triangular grip) and magnitude (circular grip).



i) Start Angle

Specify the draft angle for the start cross section.

j) Start Magnitude

Control the relative distance of the surface from the start cross section in the direction of the draft angle before the surface starts to bend toward the next cross section.

k) End Angle

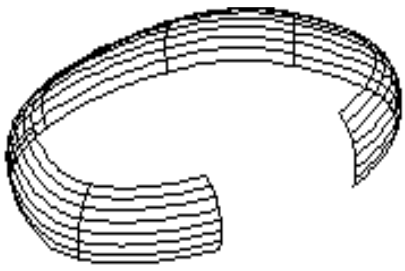
Specify the draft angle for the end cross section.

l) End Magnitude

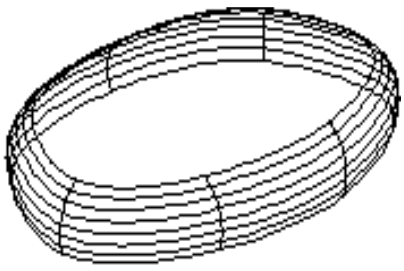
Control the relative distance of the surface from the end cross section in the direction of the draft angle before the surface starts to bend toward the previous cross section.

m) Close Surface or Solid

Close and opens a surface or solid. When using this option, the cross sections should form a torus-shaped pattern so that the lofted surface or solid can form a closed tube.



loft created with Close surface solid option unchecked



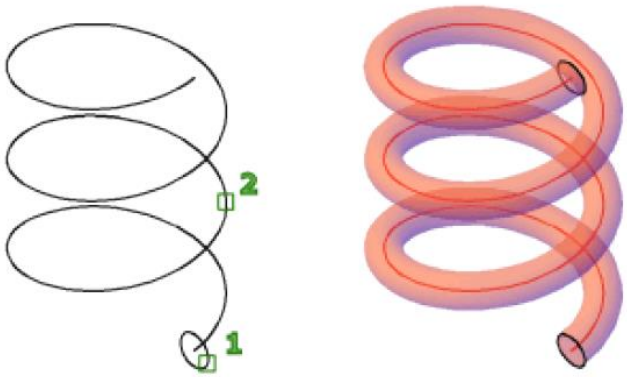
loft created with Close surface solid option checked

n) Periodic (Smooth Ends)

Create a smooth, closed surface whose seam will not kink if it is reshaped. This option is only available if the loft is ruled or smooth fit and the Close Surface or solid option is selected.

8. Sweep Surfaces

Create a solid or surface by sweeping an open or closed, planar or non-planar curve (profile) along an open or closed path. Open curves create surfaces and closed curves create solids or surfaces, depending on the specified mode. The path and the profile cannot be on the same plane.

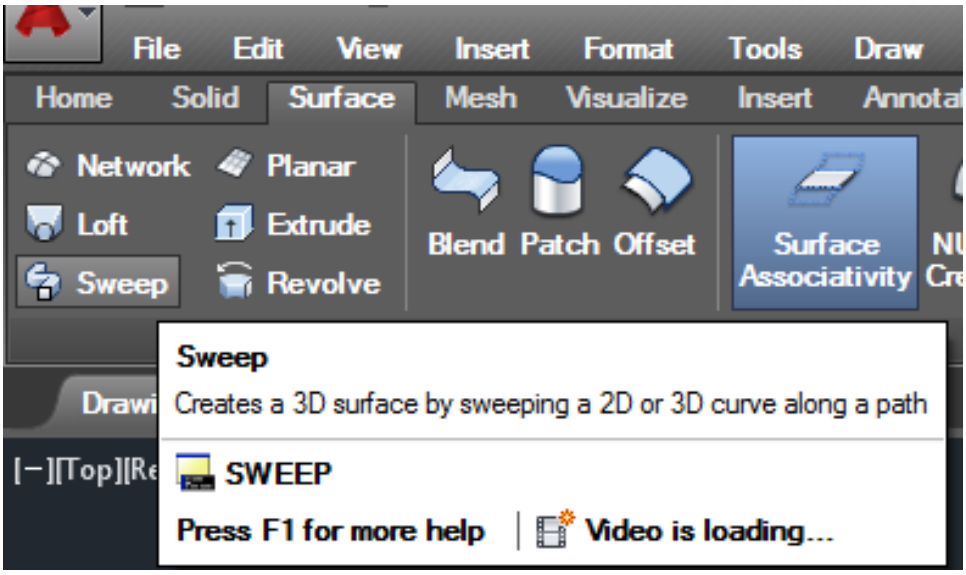


You can use the following objects and paths when creating a swept solid or surface:

Sweep Command of 3D Surface can be find out through following methods.

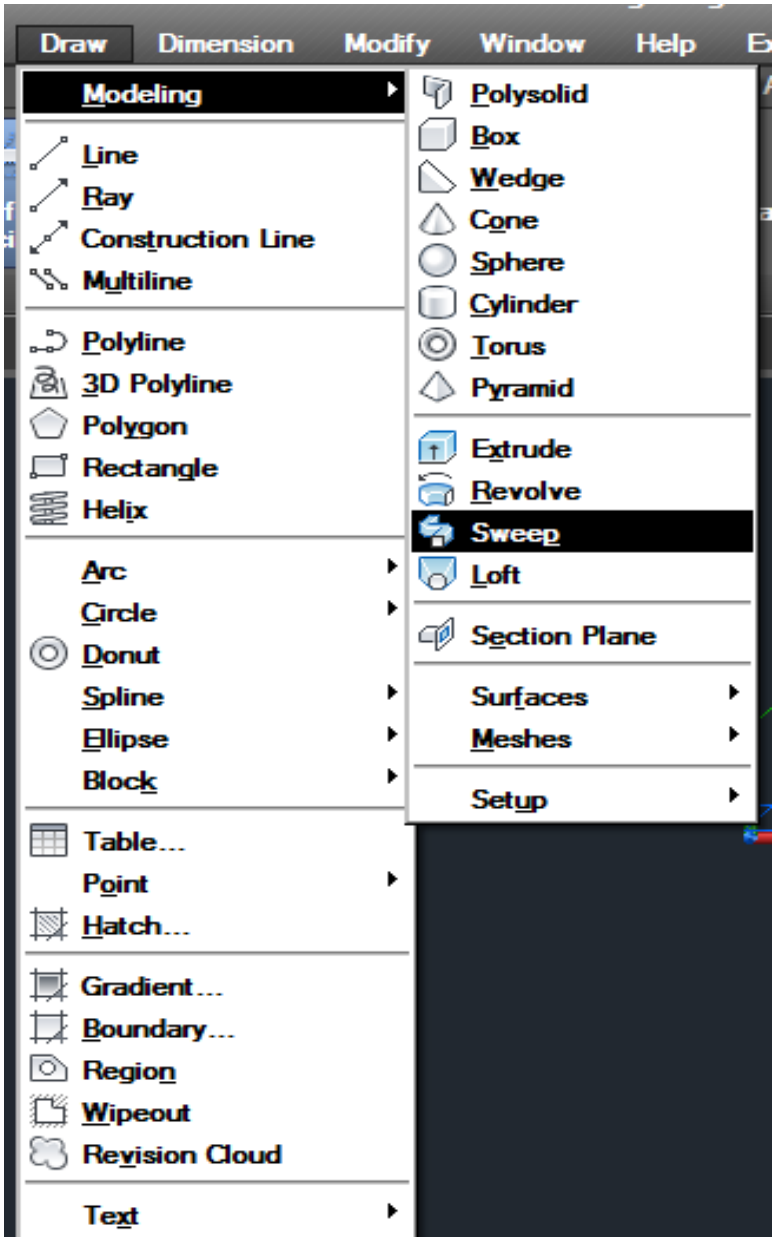
Through Ribbon

This figure describe the location of Sweep Command of 3D Surface in Surface Tab



Through Main Menu

Main Menu →Draw →Modeling →Sweep



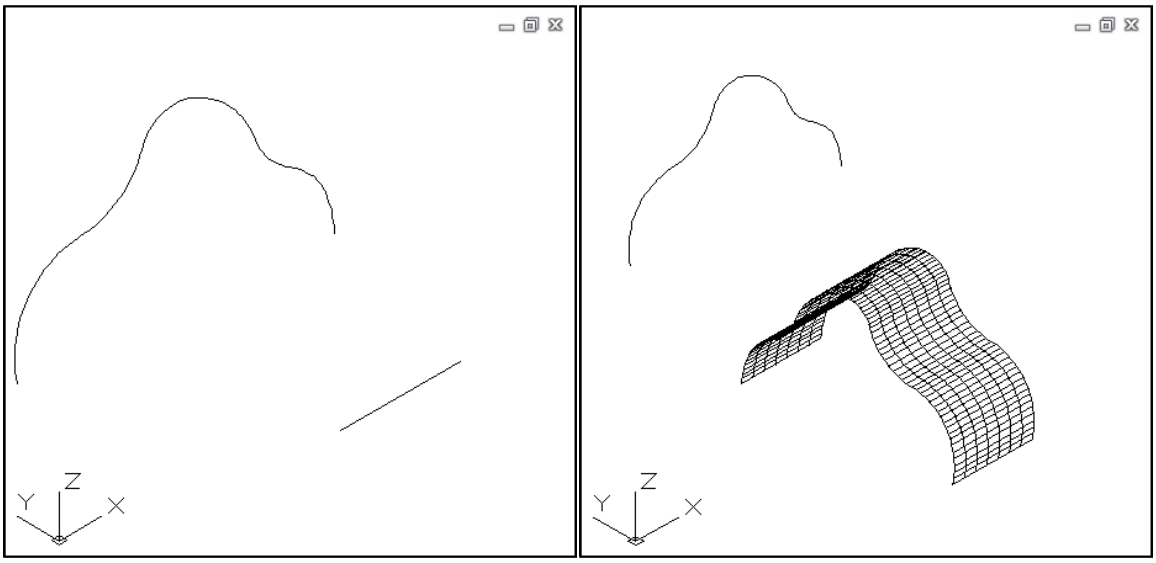
Through Short Cut Key

SWEEP → enter

APPLICATION

Type SWEEP at the Command Prompt → Select objects to sweep or [MOde]: select object→ Enter→ Select sweep path or [Alignment/Base point/Scale/Twist]: → Enter

Note:
Closed profiles creation mode = Surface



Sweep Options

i. **Alignment**

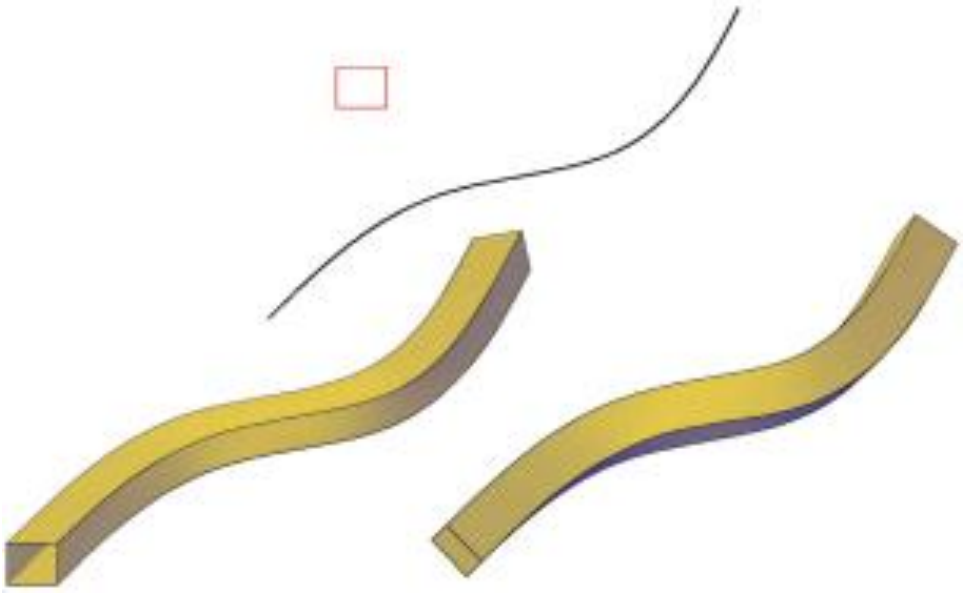
ii. **Base point**

iii. **Scale**

iv. **Twist**

Alignment

Specify whether the profile is aligned to be normal to the tangent direction of the sweep path. If the profile is not perpendicular (normal) to the tangent of the start point of the path, then the profile automatically aligns. Enter No at the alignment prompt to prevent this.



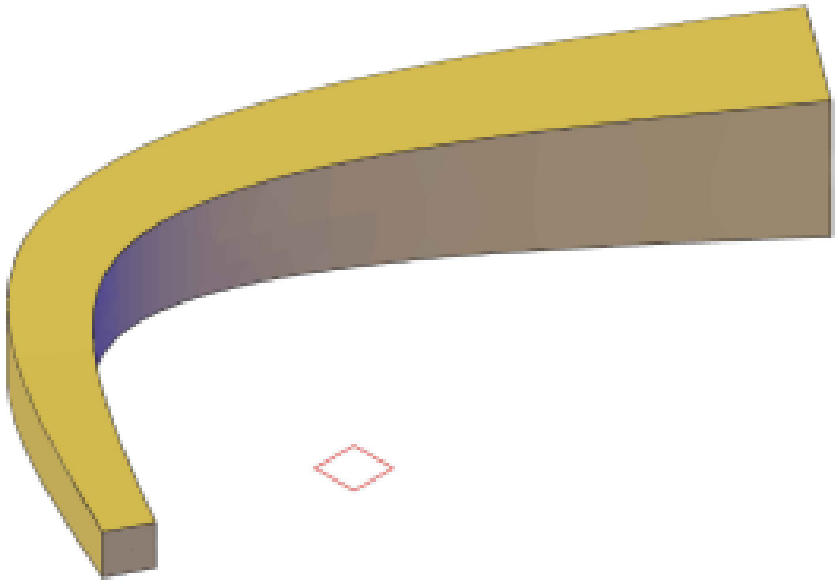
i. Base point

Specify a base point for the objects to be swept.

ii. Scale

Specify a scale factor for a sweep operation. The scale factor is uniformly applied to the objects that are swept from the start to the end of the sweep path.

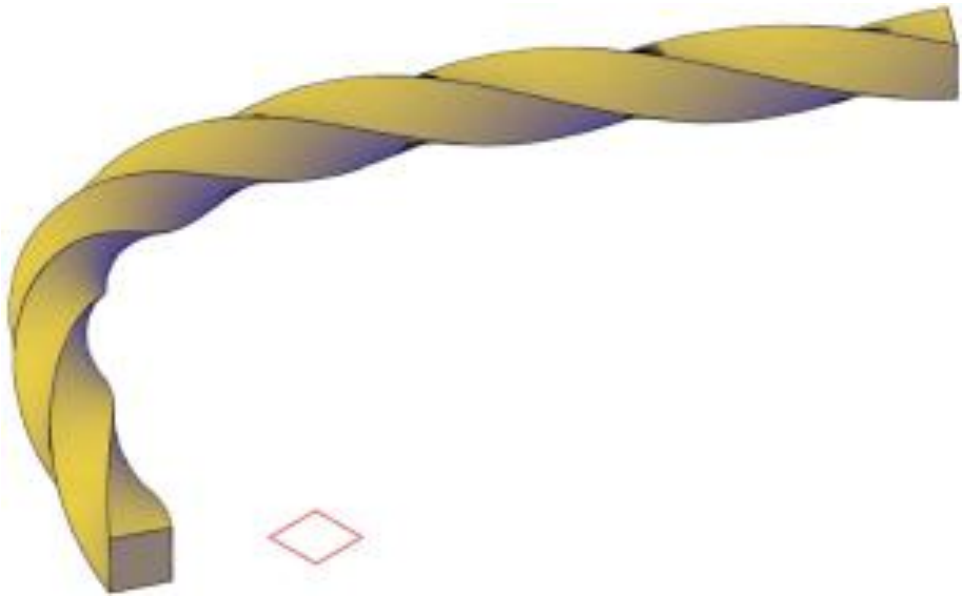
- **Reference.** Scale the selected objects based on the length you reference by picking points or entering values.



iii. Twist

Set a twist angle for the objects being swept. The twist angle specifies the amount of rotation along the entire length of the sweep path.

- **Bank.** Specify whether the curve(s) being swept will naturally bank (rotate) along a 3D sweep path (3D polyline, spline, or helix).



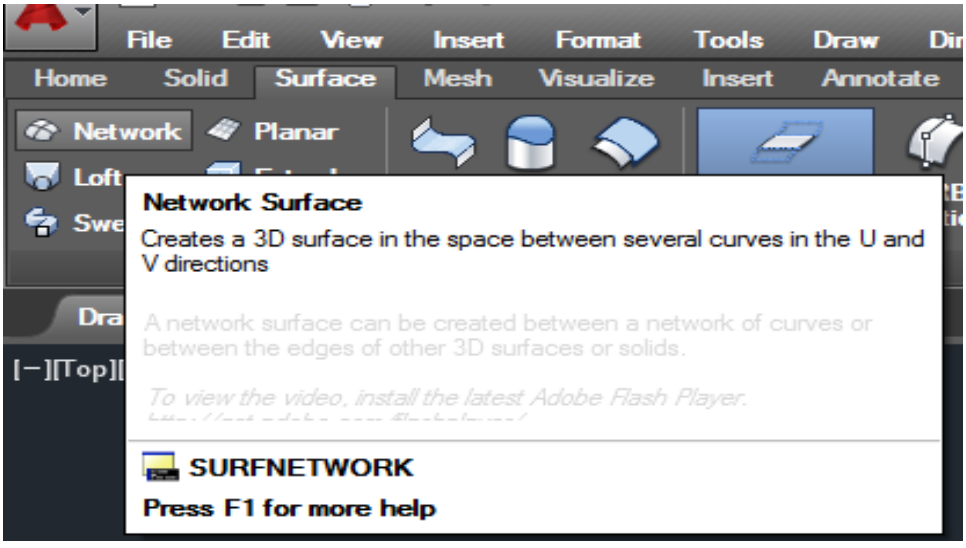
9. Surface Network

One way to create surfaces is with a network of curves. The SURFNETWORK command creates a surface similar to a lofted surface from a network of curves in the U and V directions. A network surface also can be created from curves that are not all connected. The resulting surface can be either a procedural surface or a NURBS surface.

Network Surface Command of 3D Surface can be find out through following ways

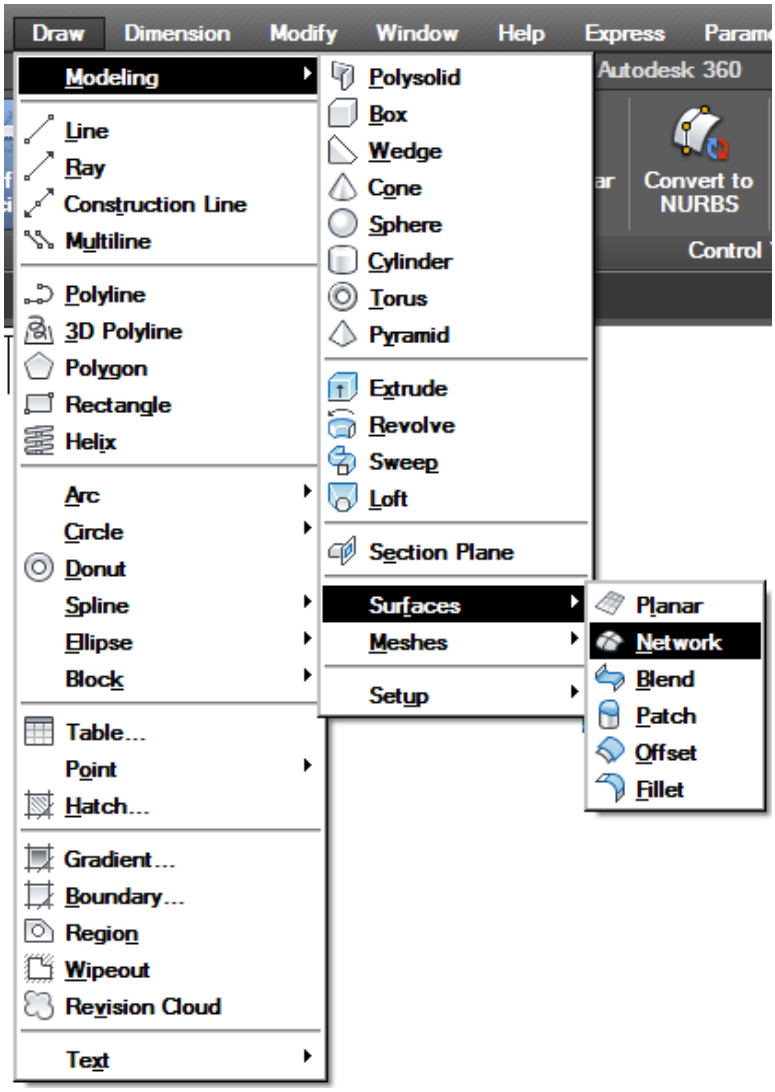
Through Ribbon

This figure describe the location of Network Surface Command of 3D Surface in Surface Tab



Through Main Menu

Main Menu →Draw →Modeling →Surfaces→Network

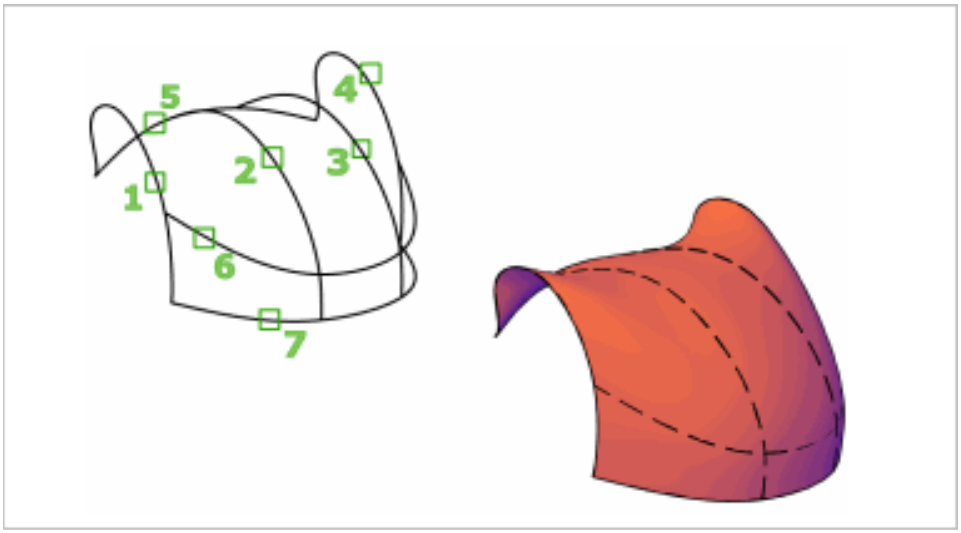


Through Short cut Key

SURFNETWORK → enter

APPLICATION

Type **SURFNETWORK** at the Command Prompt → Select Object→enter



10. Revolved Surfaces

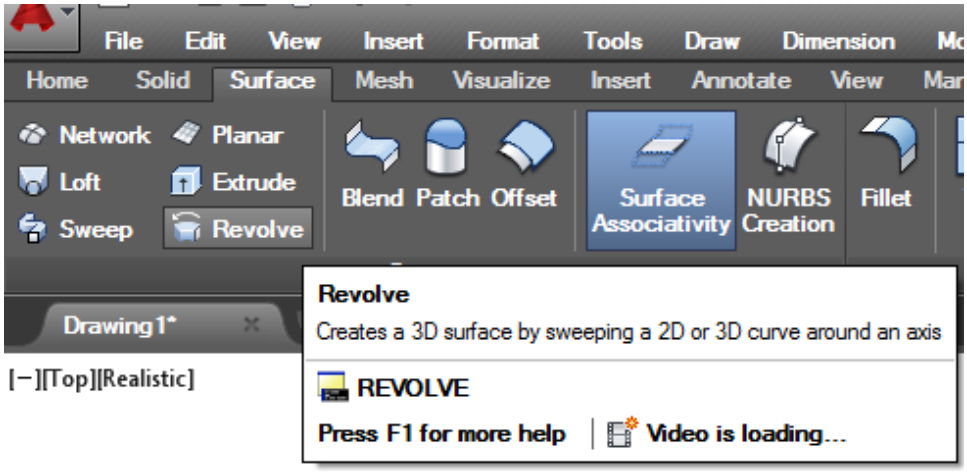
Open profile create surfaces and closed profiles can create either a solid or a surface. The Mode option controls either a solid or surface is created. When creating a surface, you cannot revolve objects contained within a block or objects that will self intersect.

REVOLVE ignores the width of a polyline and revolves from the center of the path of the polyline. The right-hand rule determines the positive direction of rotation. For more information, see Understand the UCS in 3D.

Revolve Command of 3D Surface can be find out through following methods.

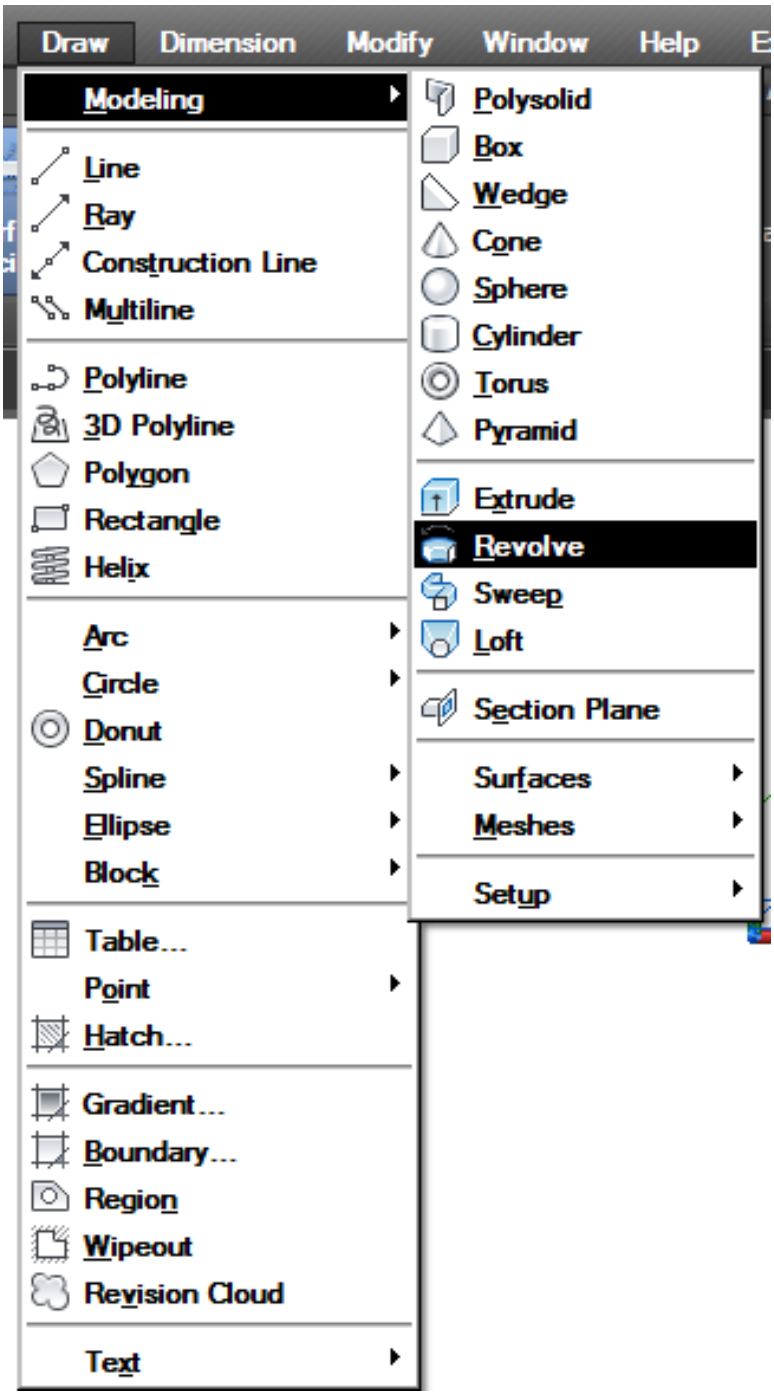
Through Ribbon

This figure describes the location of Revolve Command of 3D Surface in Surface Tab.



Through Main Menu

Main Menu →Draw →Modeling →Revolve



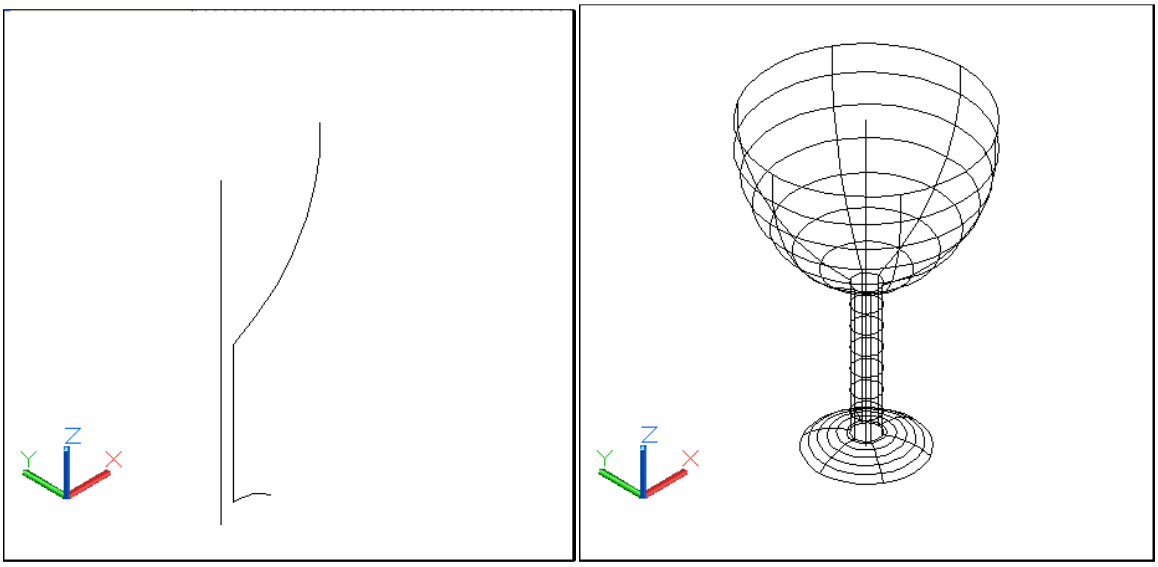
Through Short cut Key

REVOLVE → enter

APPLICATION

Type REVOLVE at the Command Prompt → Select objects to revolve or [MOde]:→enter→ Specify axis start point or define axis by [Object/X/Y/Z] Object>:→ select fist endpoints of axis →Specify 2nd axis

endpoint→ Specify angle of revolution or [Start angle/Reverse/EXpression] <360>→put the angle of revolution →enter



Revolve Options

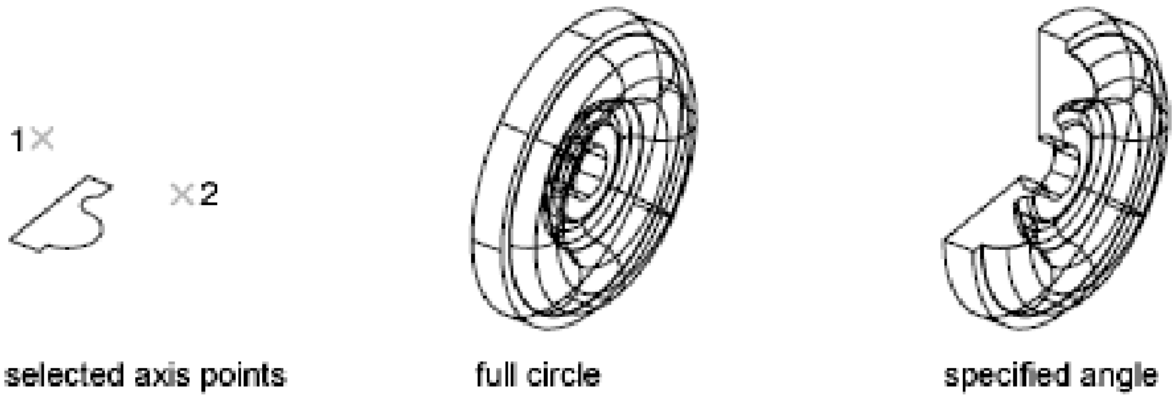
i. **Mode**
ii. **Start Angle**
iii. **Reverse**
iv. **Expression**

i. Mode

Control whether the revolve action creates a solid or a surface. Surfaces are extended as either NURBS surfaces or procedural surfaces, depending on the SURFACEMODELINGMODE system variable

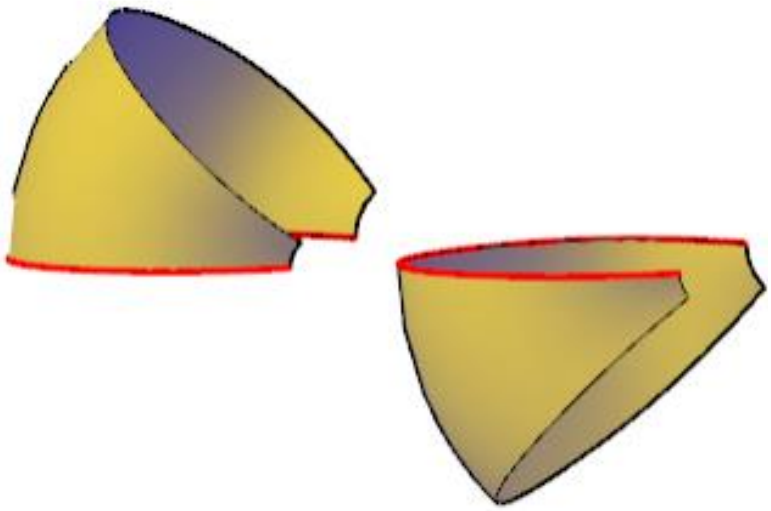
ii. Start Angle

Specify an offset for the revolution from the plane of the object being revolved. Drag your cursor to specify and preview the start angle of the object.



iii. Reverse

Change the direction of the Revolve; similar to entering a - (minus) angle value. The revolved object on the right shows a spline revolved at the same angle as the object on the left, but using the reverse option.



iv. Expression

Enter a formula or equation to specify the revolve angle. See Constrain a Design with Formulas and Equations.

11. Surface Blend

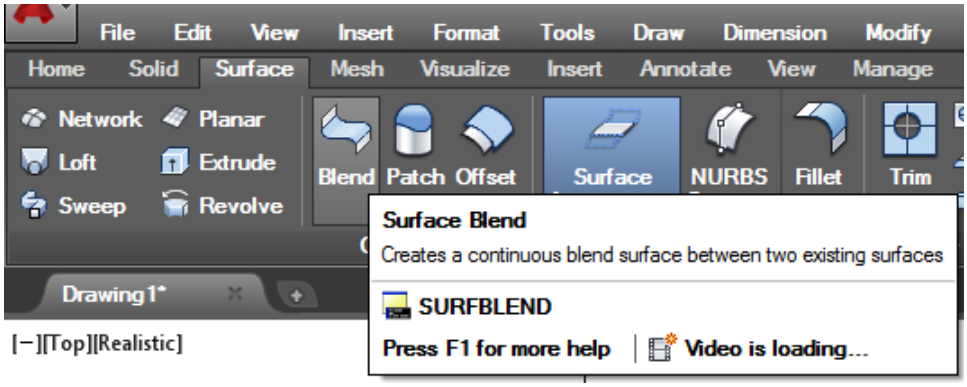
Create a continuous blend surface between two existing surfaces. When you blend two surfaces together, you can specify surface continuity and bulge magnitude.

Use SURFBLEND to create a new surface between existing surfaces and solids. When you blend surfaces together, specify the surface continuity and bulge magnitude for the start and end edges.

Surface Blend Command of 3D Surface can be find out through following methods.

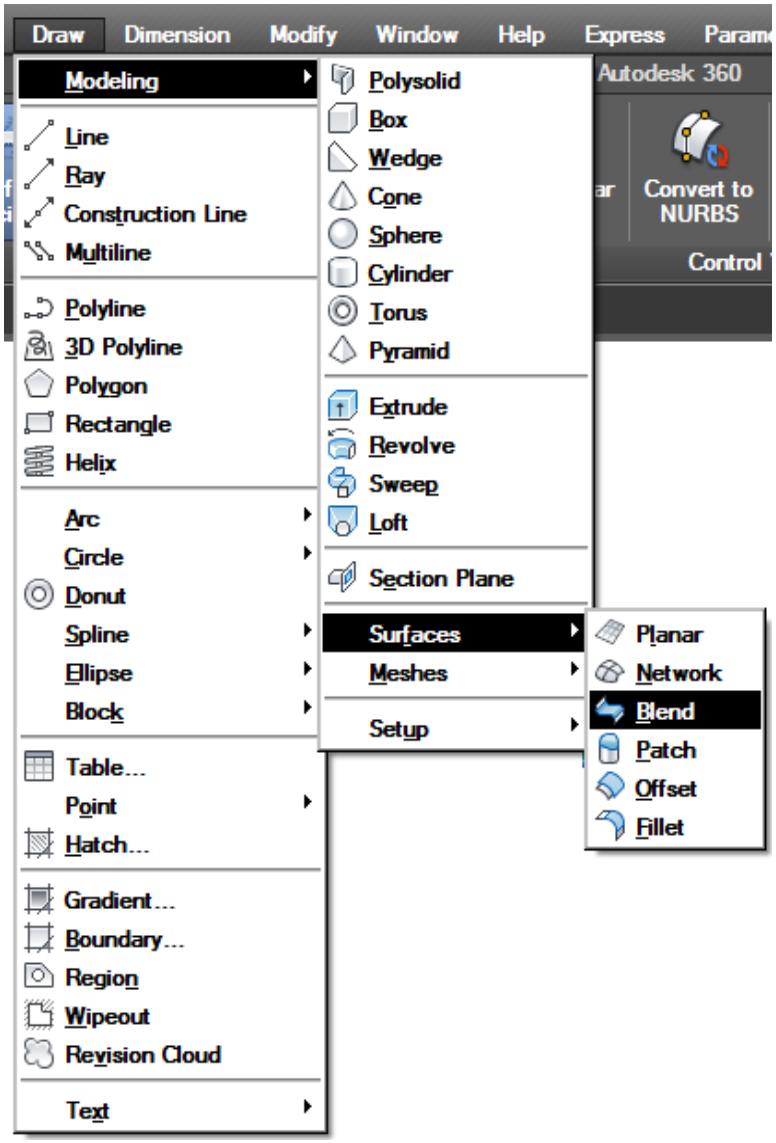
Through Ribbon

This figure describe the location of Surface Blend Command of 3D Surface in Surface Tab



Through Main Menu

Main Menu →Draw →Modeling →Surfaces→Blend

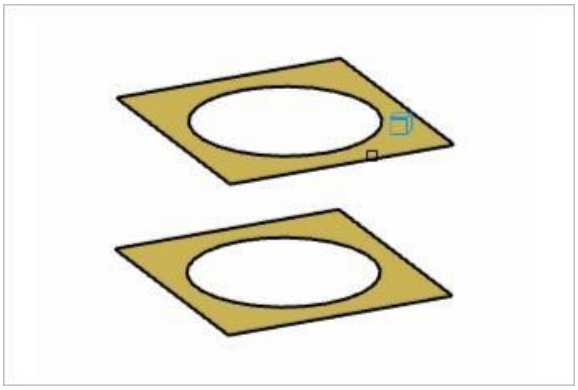


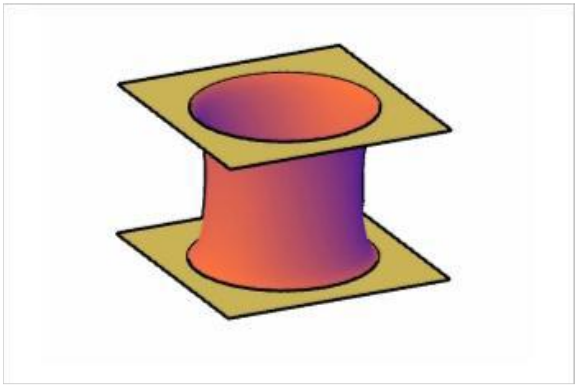
Through Short Cut Key

SURFBLEND → enter

APPLICATION

Type SURFBLEND at the Command Prompt → Select first edge of object to blind → Select 2nd edge of object to blind → enter





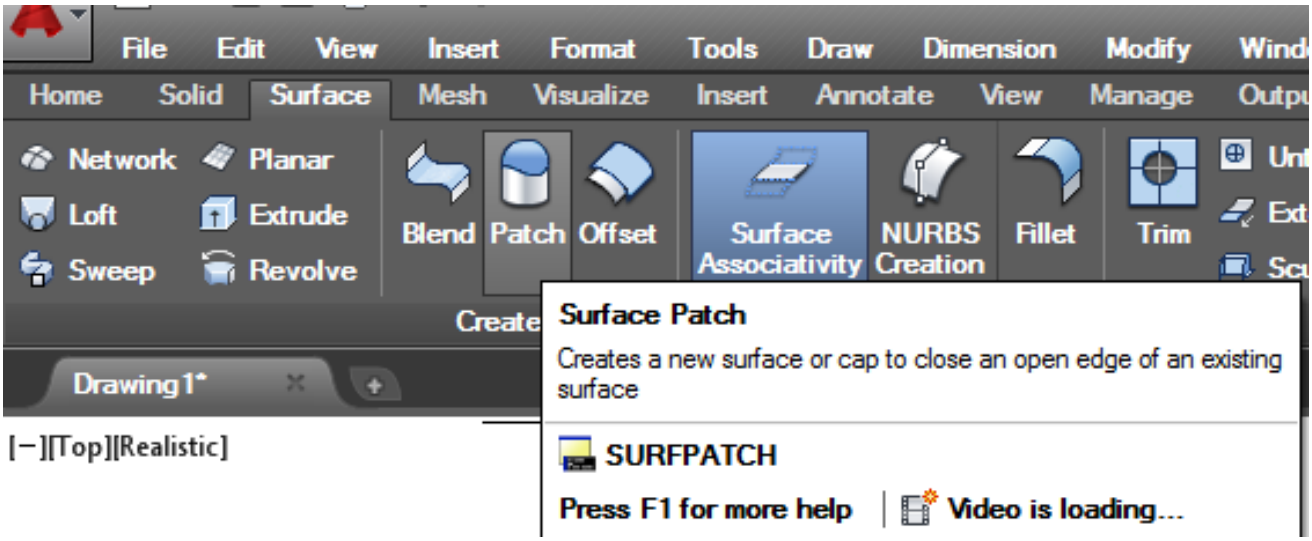
12. Surface Patch

Create a surface by patching a closed surface or curve. Use SURFPATCH to create a surface inside a closed curve (such as a closed spline) that is another surface's edge. You can also draw a guide curve to constrain the shape of the patch surface with the constrain geometry option. When you patch surfaces, specify the continuity and bulge magnitude.

Surface Patch Command of 3D Surface can be find out through following methods.

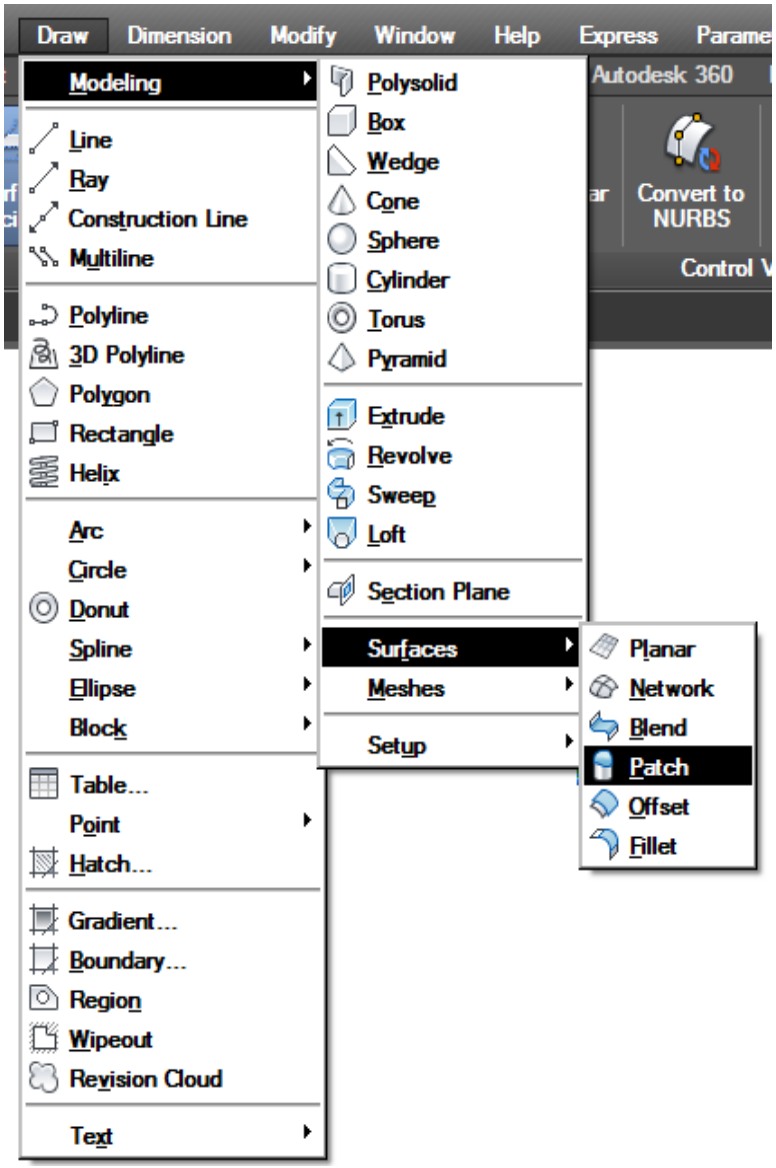
Through Ribbon

This figure describes the location of Surface Patch Command of 3D Surface in Surface Tab.



Through Main Menu

Main Menu →Draw →Modeling →Surfaces→Patch

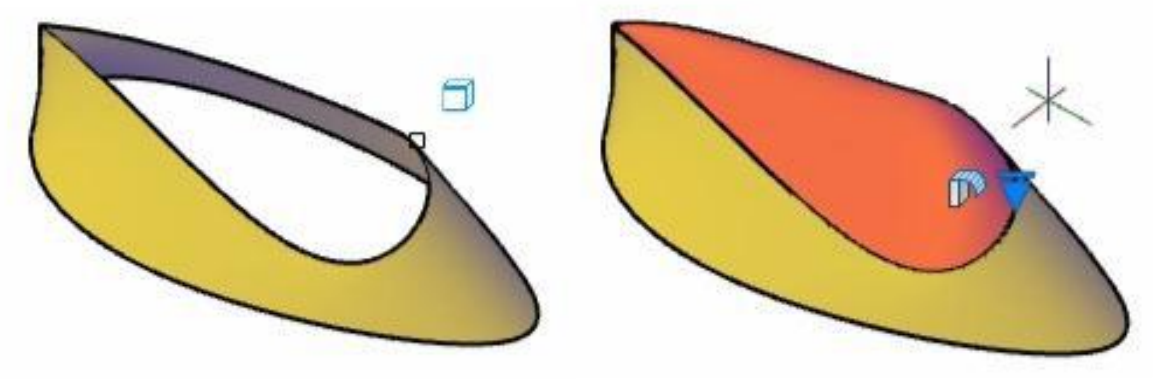


Through Short Cut Key

SURFPATCH → enter

APPLICATION

Type SURFPATCH at the Command Prompt → Select edge of object to patch → enter



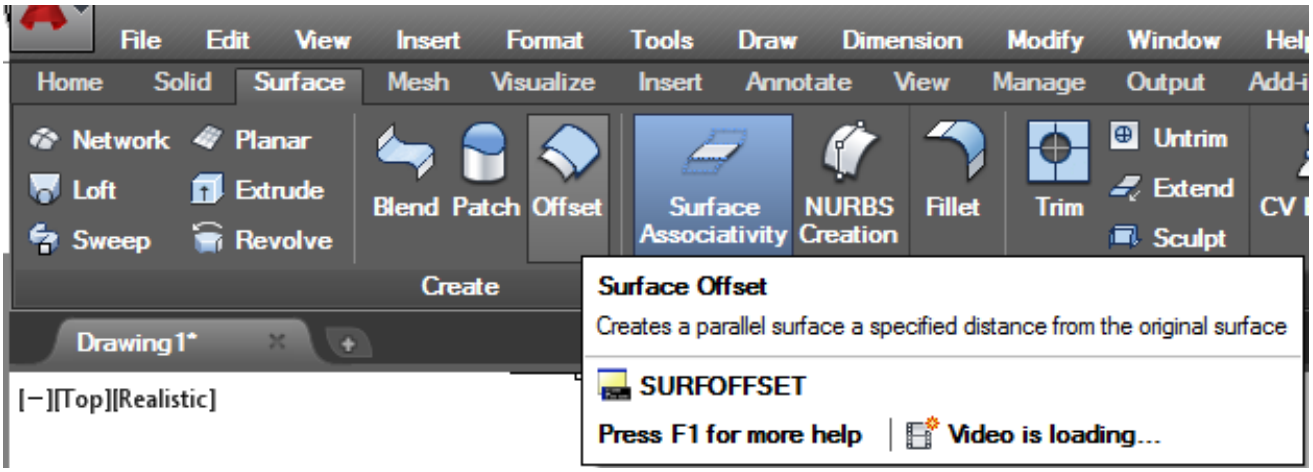
13. Offset Surfaces

Create a parallel surface to a specified distance from the original surface. With SURFOFFSET specify the offset distance and whether or not the offset surface maintains associatively with the original surface. You can also specify the offset distance with a mathematical expression.

Offset Surface Command of 3D Surface can be find out through following ways

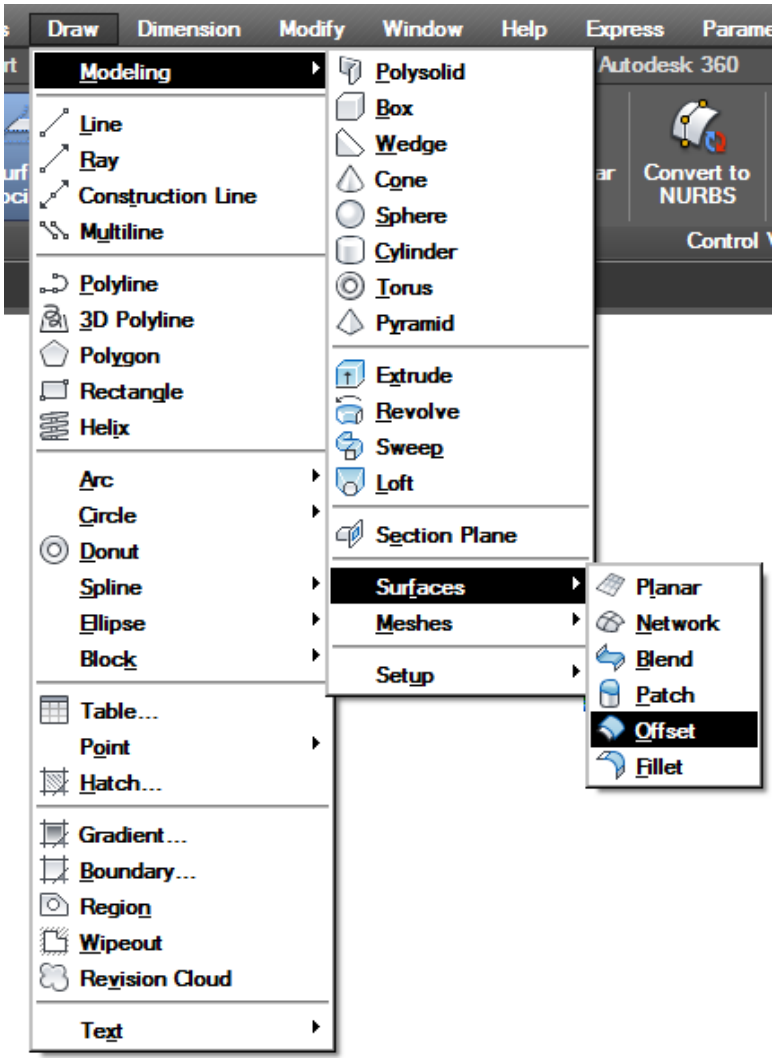
Through Ribbon

This figure describe the location of Offset Surface Command of 3D Surface in Surface Tab



Through Main Menu

Main Menu →Draw →Modeling →Surfaces→Offset

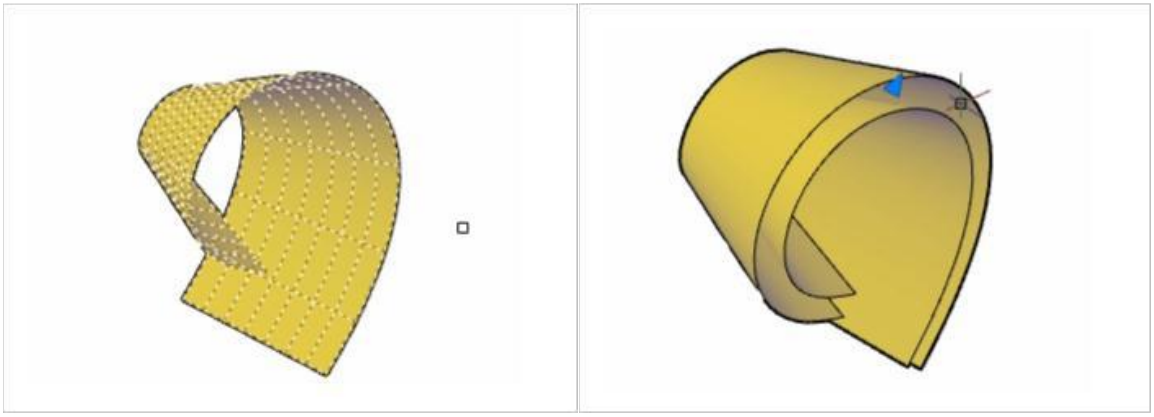


Through Short cut Key

SURFOFFSET → enter

APPLICATION

Type SURFOFFSET at the Command Prompt → Select surfaces or regions to offset → Specify offset distance or [Flip direction/both sides/Solid/Connect/Expression] →Put the value of offset→enter



Surface Offset Options

i. **Flip direction**

ii. **Both sides**

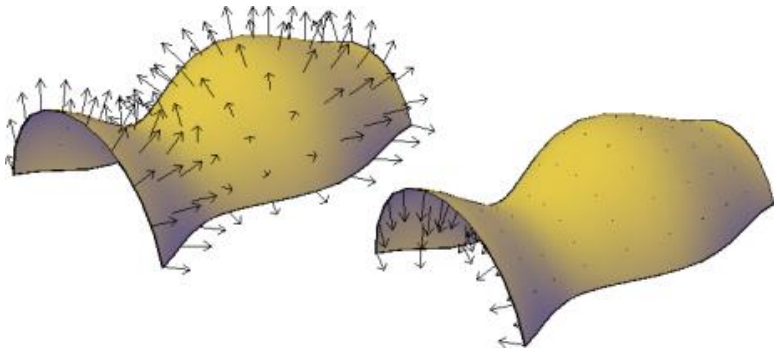
iii. **Solid**

iv. **Connect**

v. **Expression**

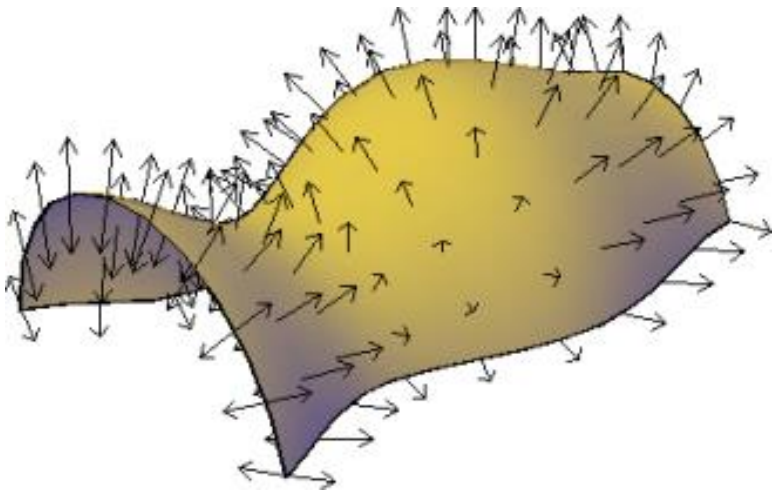
i. Flip direction

Reverse the offset direction shown by the arrows.



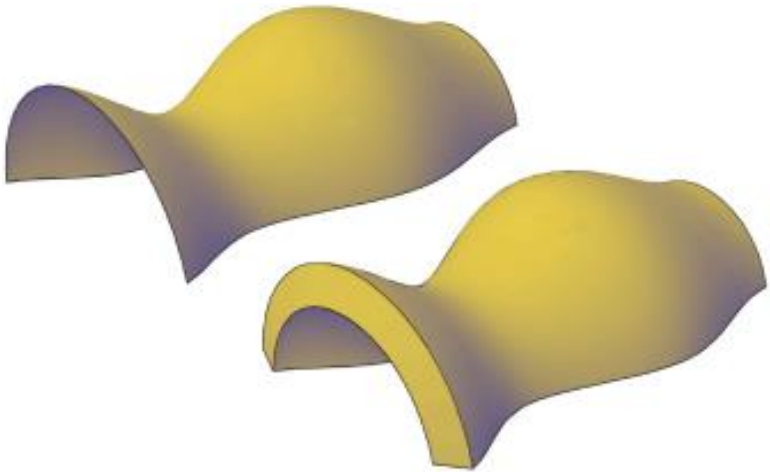
ii. Both sides

Offset the surface in both directions (create two new surfaces in instead of one).



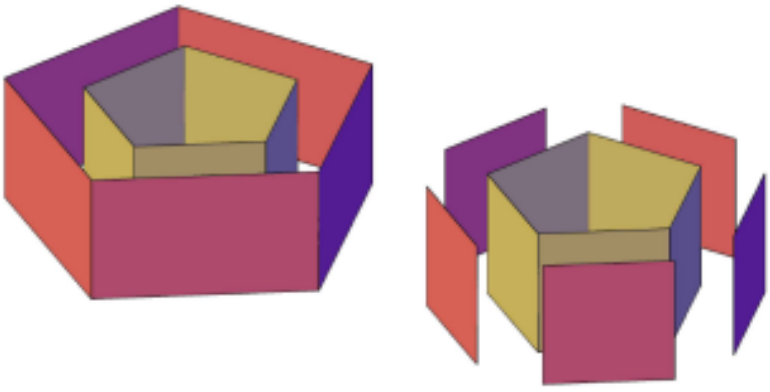
iii. Solid

Create a solid from the offset. This is similar to the THICKEN command.



iv. Connect

Connect multiple offset surfaces, if the original surfaces are connected.



v. Expression

Enter a formula or equation to specify the distance of the surface offset. See Constrain a Design with Formulas and Equations.

14. Editing Surfaces

The following commands use the editing surface objects.

1. **Fillet**
2. **Trim**
3. **Untrim**
4. **Extend**
5. **Sculpt**

6. **Extract Intersections**

7. **Offest Edge**

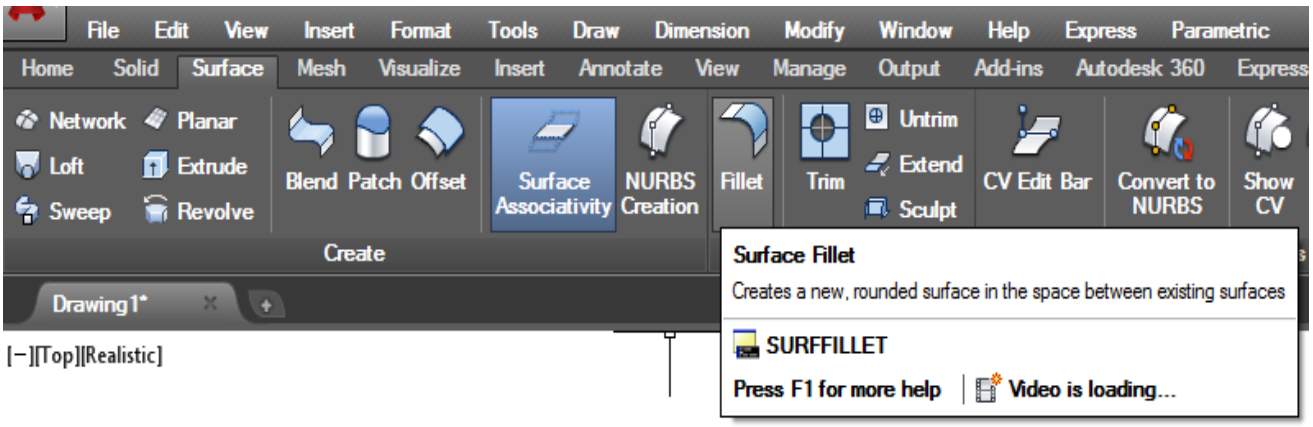
1) Surface Fillet

The fillet surface has a constant radius profile and is tangent to the original surfaces. The original surfaces are automatically trimmed to connect the edges of the fillet surface.

Surface Fillet Command of 3D Surface can be find out through following methods.

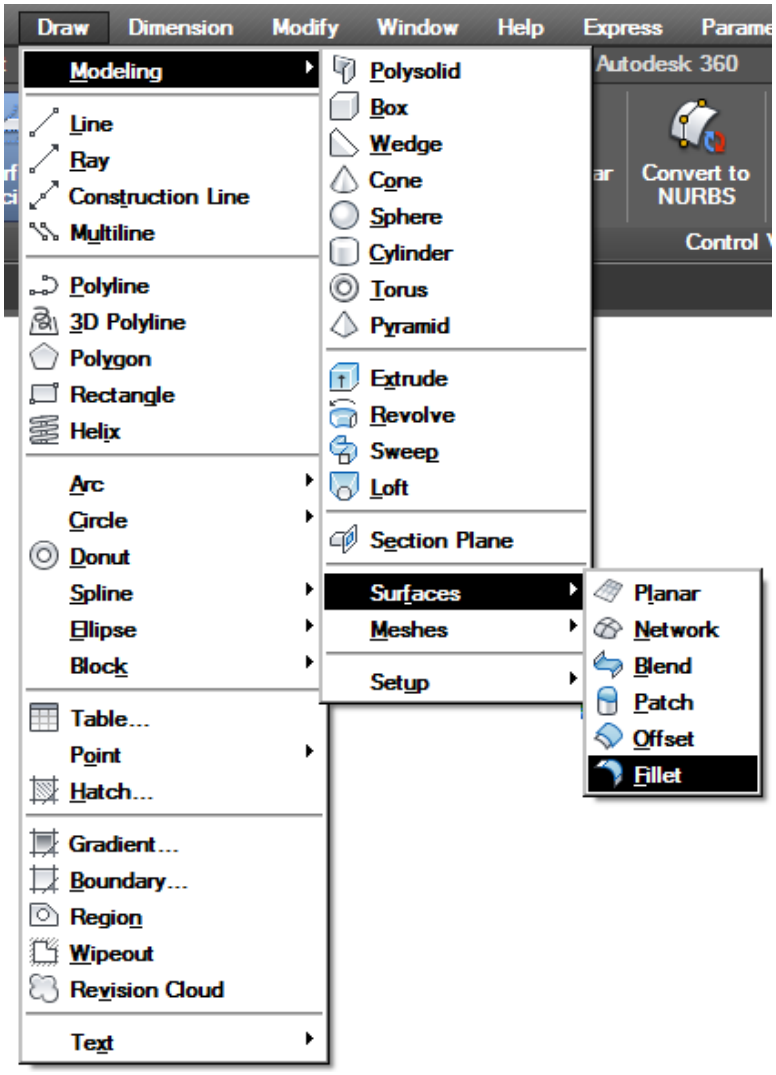
Through Ribbon

This figure describe the location of Surface Fillet Command of 3D Surface in Surface Tab



Through Main Menu

Main Menu →Draw →Modeling →Surfaces→Fillet

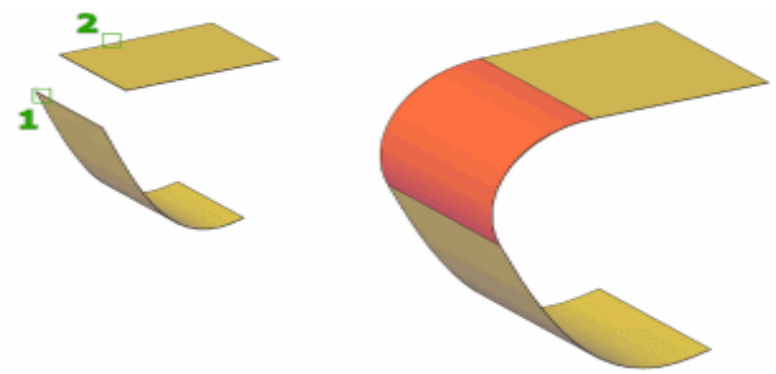


Through Short Cut Key

SURFFILLET → enter

APPLICATION

Type SURFFILLET at the Command Prompt → Select first surface or region to fillet or [Radius/Trim surface]:→ R enter (for Radius)→ Put the Radius Value→Select first Surface→Select 2nd surface→enter



Surface Fitted Option

i. Radius

Specify fillet radius. Use the Fillet grip or enter a value to change the radius. You cannot enter a value that is smaller than the gap between the surfaces. If no radius value is entered,

ii. Trim Surface

Trim the original surfaces or regions to the edges of the fillet surface.

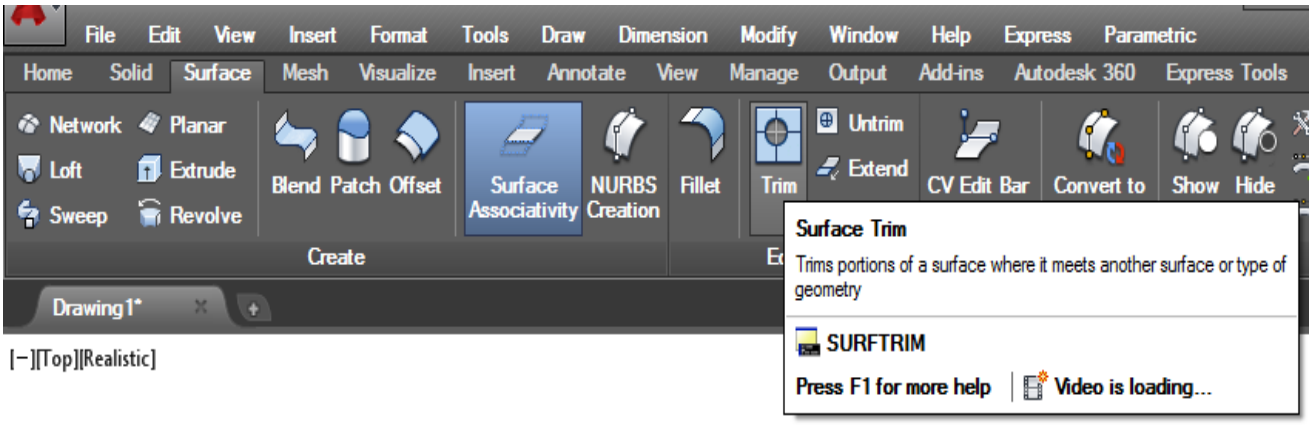
2) Surface Trim

Trim surface to meet the edges of other objects. An important step in the surface modeling workflow is trimming surfaces. You can trim a surface where it meets an intersecting object or you can project geometry onto a surface as a trimming edge. When a surface is trimmed, the removed surface areas can be replaced with SURFUNTRIM

Surface Trim Command of 3D Surface can be find out through following methods

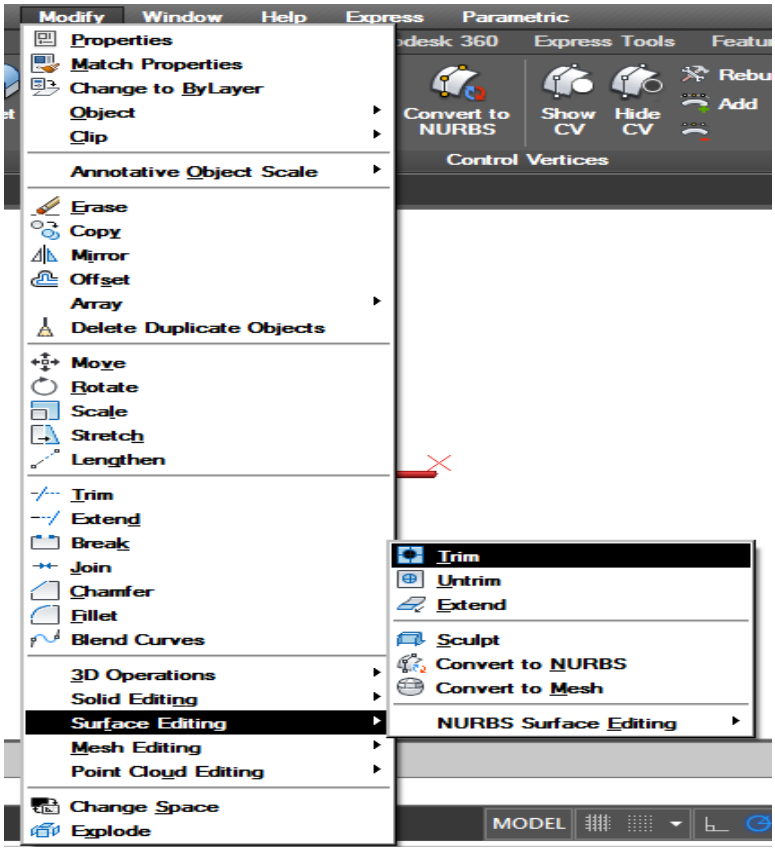
Through Ribbon

This figure describe the location of Surface Trim Command of 3D Surface in Surface Tab



Through Main Menu

Main Menu →Modify →Surface Editing→Trim

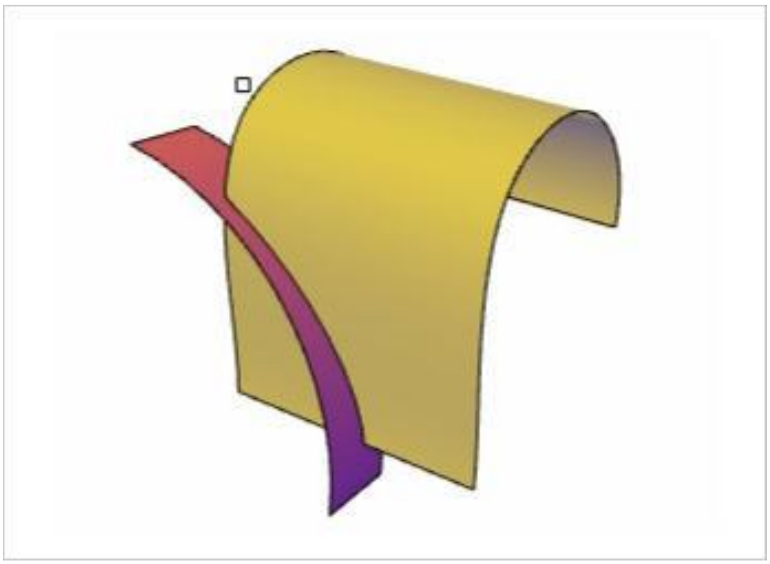


Through Short Cut Key

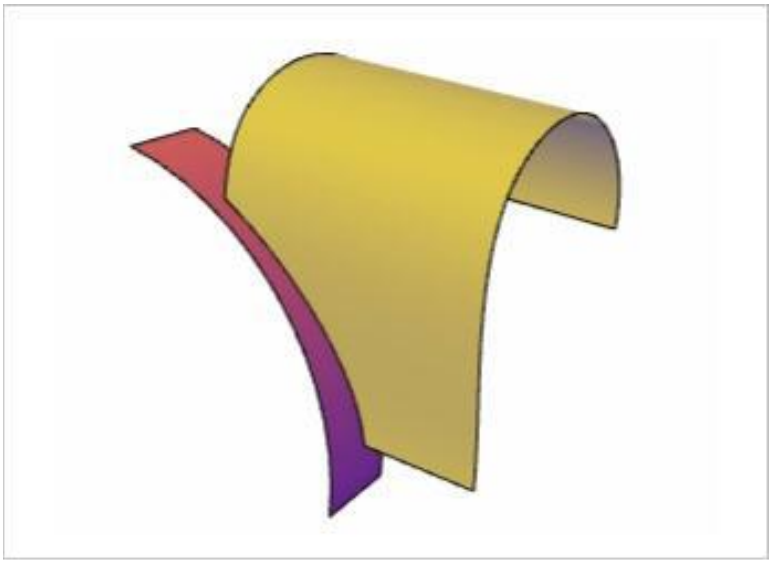
SURFTRIM → enter

APPLICATION

Type SURFTRIM at the Command Prompt → Select cutting curves, surfaces or regions → Select area to trim → enter



Note: SURFUNTRIM does not restore areas removed by the SURFAUTOTRIM System variable and PROJECTGEOMETRY. It only restores areas



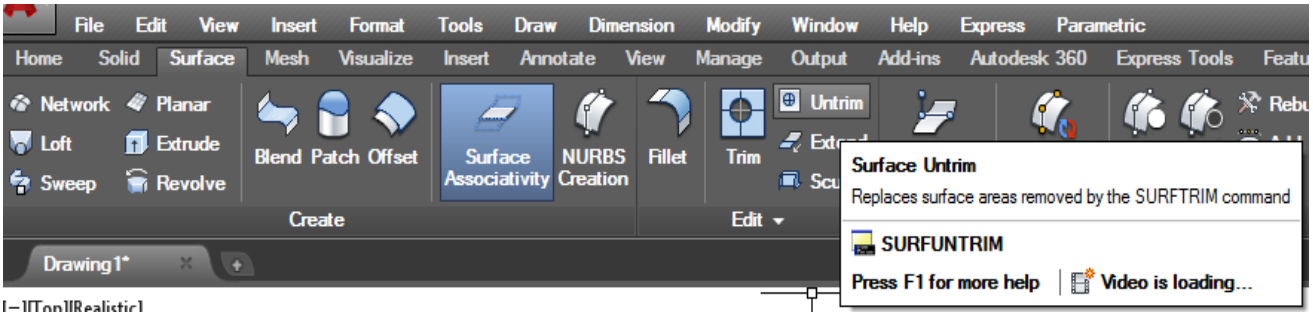
3) Surface UnTrim

Replace surface areas removed by the SURFTRIM command. If the trimmed edge is dependent on another surface edge that has also been trimmed, you may not be able to fully restore the trimmed area.

Surface Untrim Command of 3D Surface can be find out through following methods.

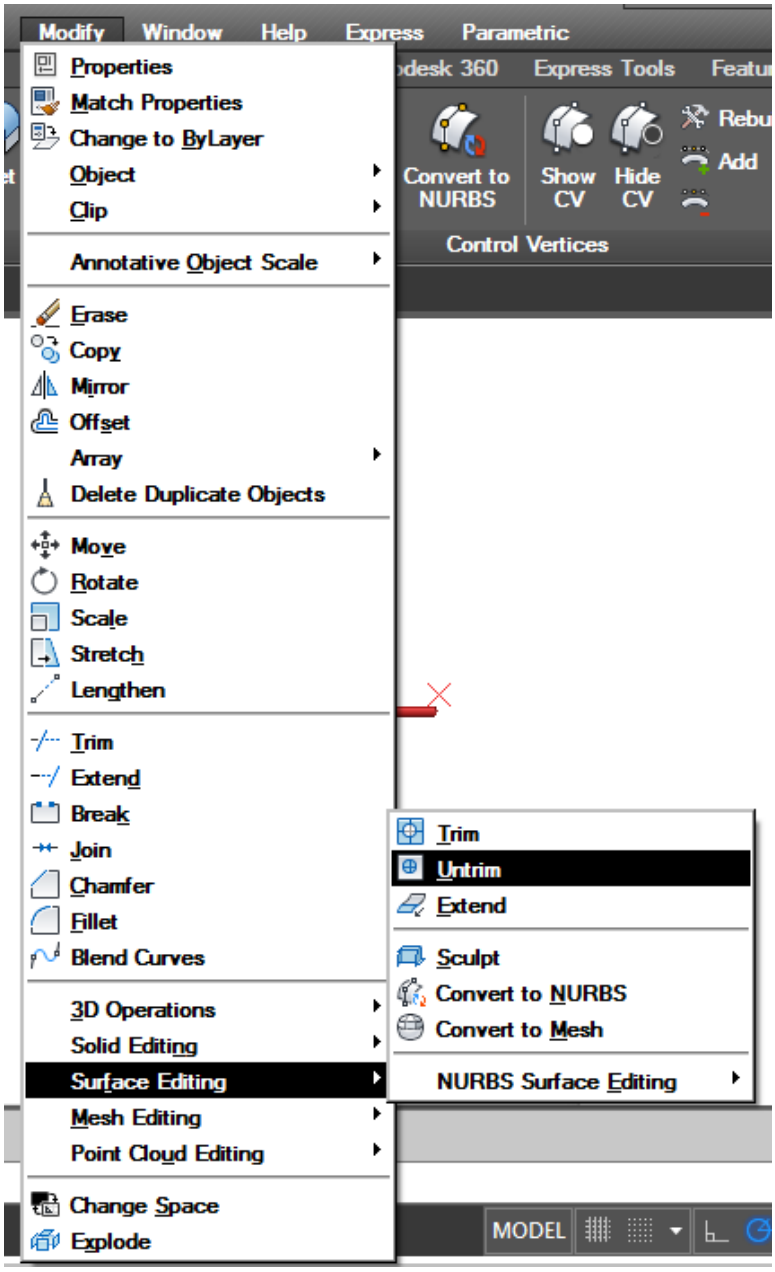
Through Ribbon

This figure describe the location of Surface Untrim Command of 3D Surface in Surface Tab



Through Main Menu

Main Menu →Modify → Surface Editing→Untrim



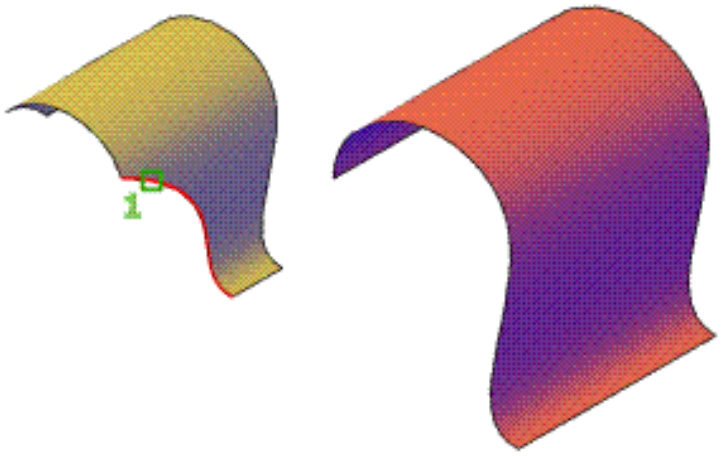
Through Short Cut Key

SURFUNTRIM → enter

APPLICATION

Type SURFUNTRIM at the Command Prompt → Select cutting curves, surfaces or regions → enter

Note:
SURFUNTRIM does not restore areas removed by the SURFAUTOTRIM System variable and PROJECTGEOMETRY.



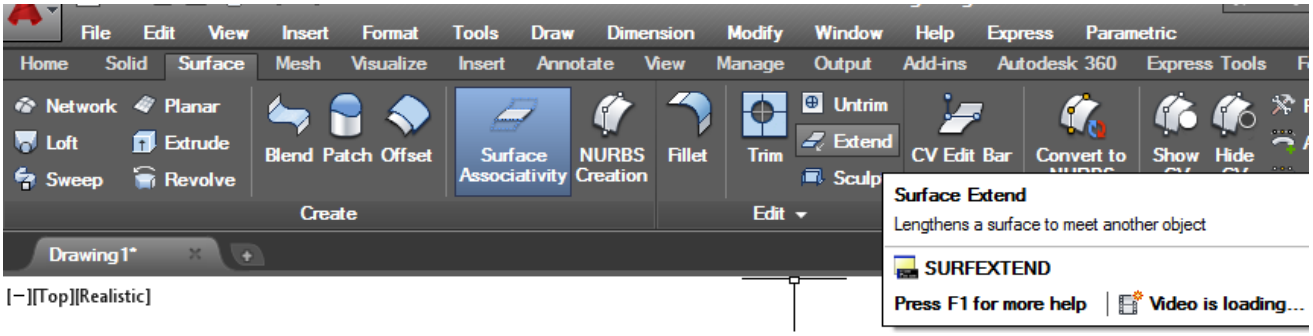
Surface Extend

Create a new surface by extending it to meet the edge of another object or by specifying an extension length

Surface Extend Command of 3D Surface can be find out through following methods.

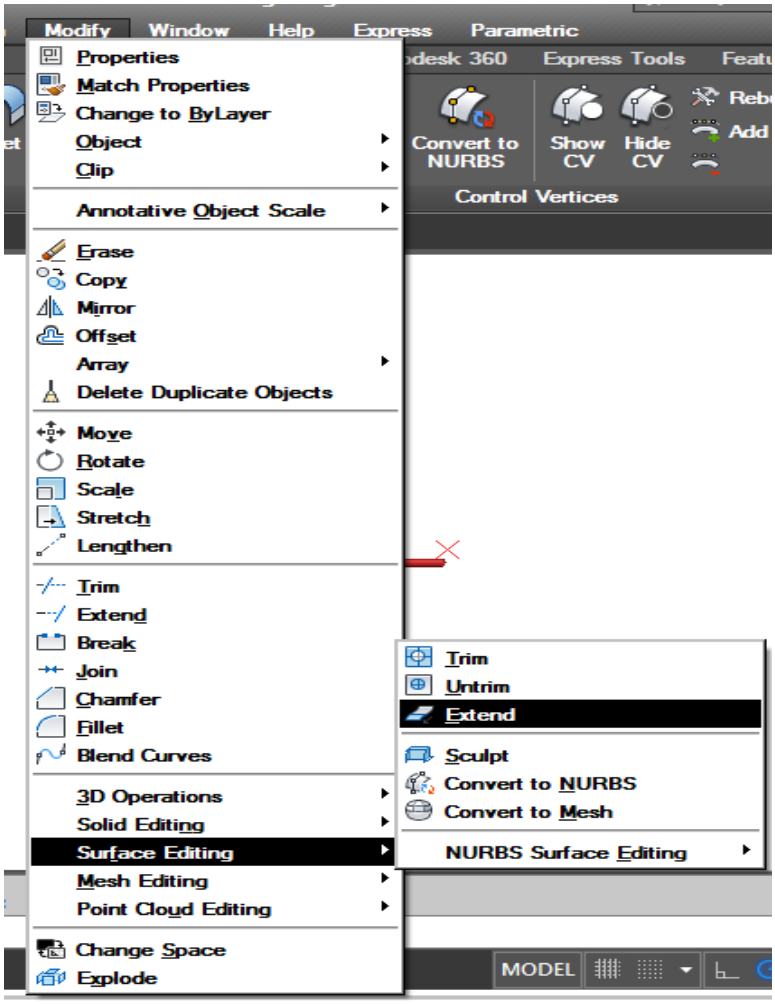
Through Ribbon

This figure describe the location of surface Extend Command of 3D Surface in Surface Tab



Through Main Menu

Main Menu →Modify →Surface Editing→Extend

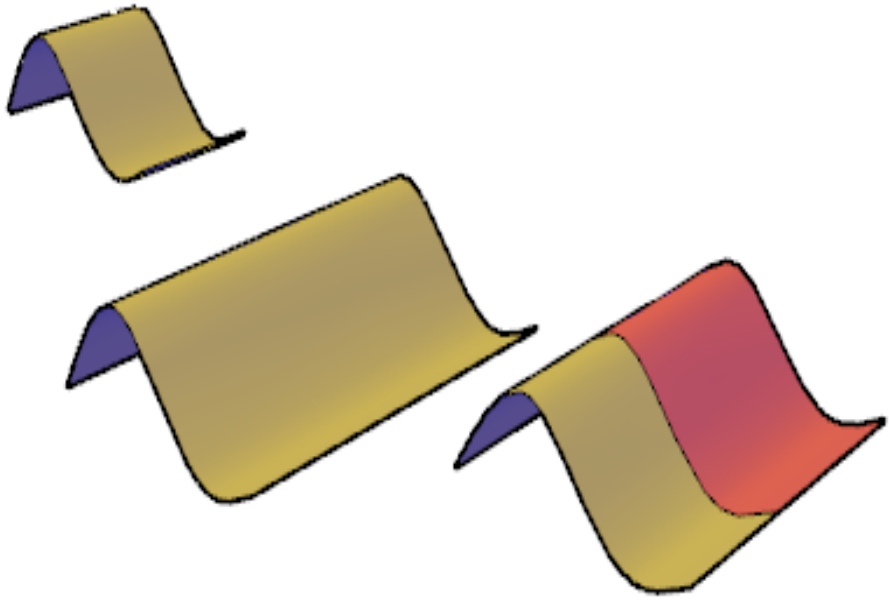


Through Short cut Key

SURFEXTEND → enter

APPLICATION

Type SURFEXTEND at the Command Prompt → Select surface edges to extend → select edge→ put the value of extension



Extend Options.

i. Merge

The merge surface is a continuation of the surface with no seam.

ii. Append.

The append surface extends the surface by adding a second surface with a seam.

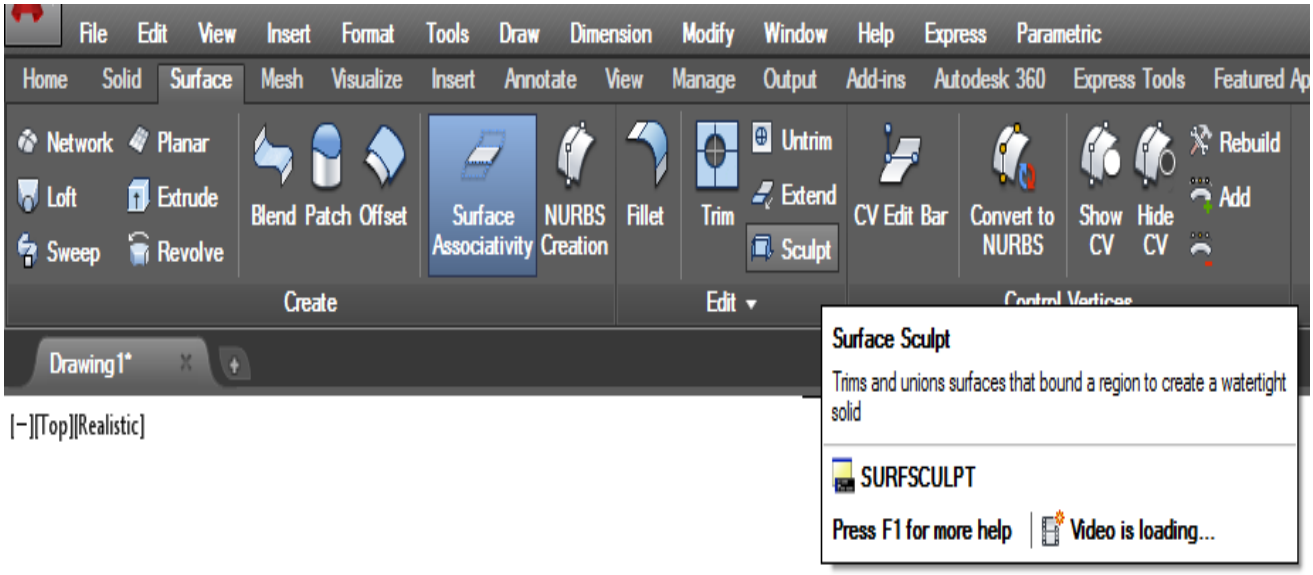
4) Surface Sculpt

The SURFSCULPT command automatically combines and trims a collection of surfaces that enclose a watertight area to create a solid. The SURFSCULPT command also works with solid and mesh objects.

Surface Sculpt Command of 3D Surface can be find out through following methods.

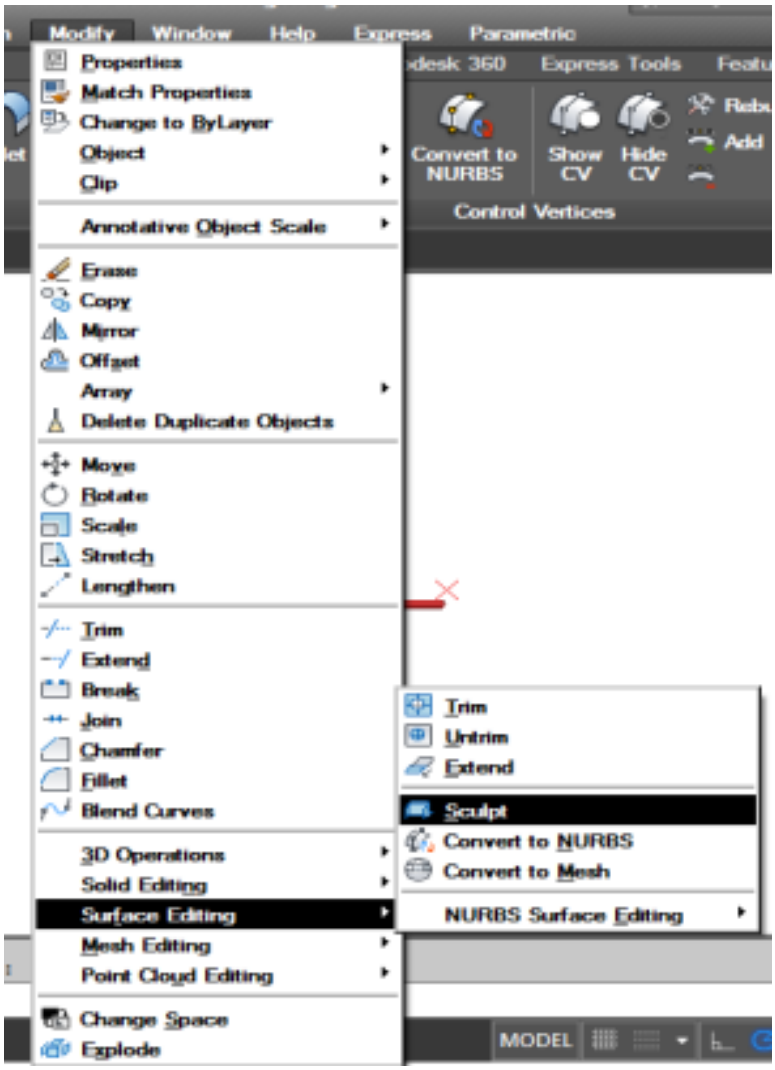
Through Ribbon

This figure describes the location of Surface Sculpt Command of 3D Surface in Surface Tab.



Through Main Menu

Main Menu →Modify → Surface Editing→Sculpt



Through Short cut Key

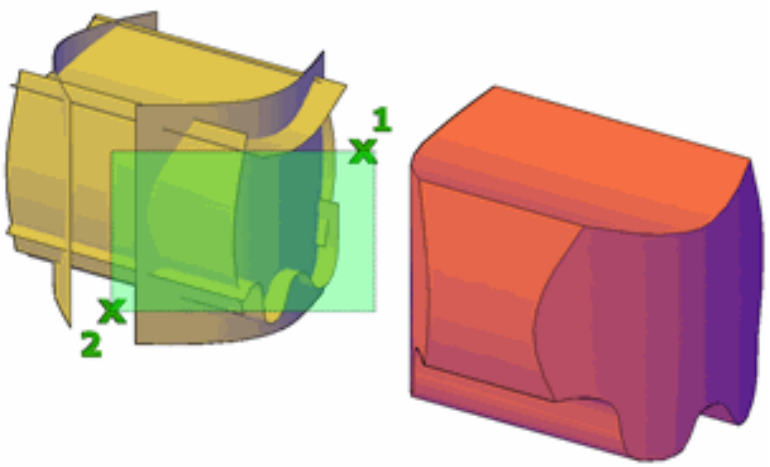
SURFSCULPT → enter

APPLICATION

Type SURFSCULPT at the Command Prompt → Select surfaces or solids to sculpt into a solid →enter

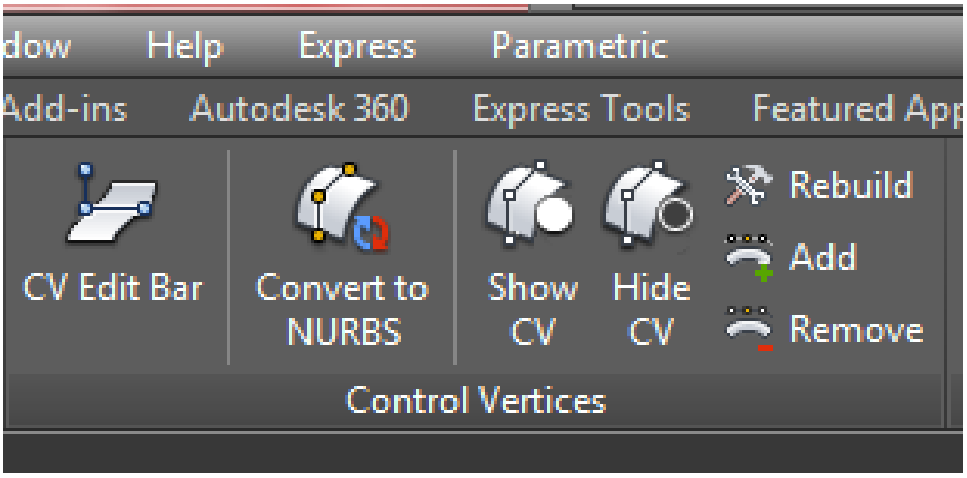
Remember

The area enclosed by the surfaces must be watertight and the surfaces must have a continuity of G0 or the SURFSCULPT command cannot be completed.



15. NURBS Surfaces

The following commands are used in Control Vertices of surface objects. It adds and edits control vertices on a NURBS surface or spline.



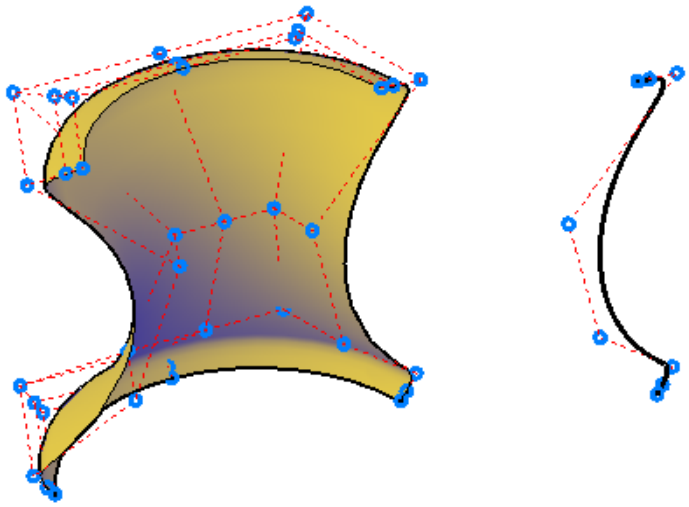
NURBS

Non-Uniform Rational B-Splines, are mathematical representations of 3-D geometry that can accurately describe any shape from a simple 2-D line, circle, arc, or curve to the most complex 3-D organic free-form surface or solid. Because of their flexibility and accuracy, NURBS models can be used in any process from illustration and animation to manufacturing.

Creating NURBS Surfaces

Create NURBS surfaces when you need a freeform way to sculpt a surface shape. Create NURBS surfaces by enabling NURBS creation and using many of the same commands used to create procedural surfaces. You can also convert existing procedural surfaces into NURBS surfaces.

NURBS surfaces are based on Bezier curves or splines. Therefore, settings such as degree, fit points, control vertices, weights, and knot parameterization are important in defining a NURBS surface or curve. Splines are optimized to create NURBS surfaces, allowing you to control many of these options. The illustration below shows the control vertices that are displayed when you select a NURBS surface or spline.



Note:
NURBS mean

Non-Uniform
Rational B-Splines

Creating NURBS Surfaces

There are Two Methods for creating NURBS Surfaces.

i. **SURFACEMODELINGMODE system variable**.

Use any of the surface creation commands while this system variable is set to 1.

ii. CONVTONURBS command.

Convert any existing procedural surfaces to NURBS surfaces with this command.

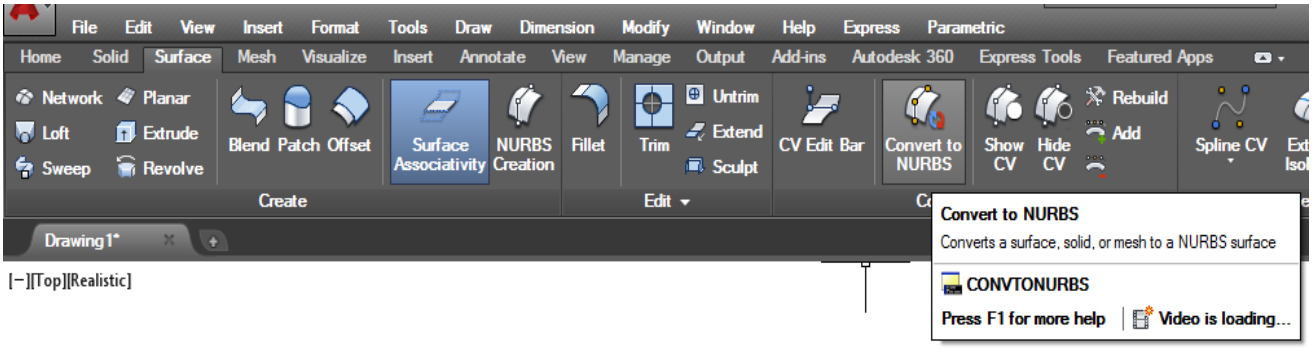
Convert To NURBS

Convert solids and procedural surfaces to NURBS surfaces. To convert meshes to NURBS surfaces, convert them to a solid or surface first with CONVTOSOLID or CONVTOSURFACE and then convert them to NURBS surfaces.

Convert to NURBS Command of 3D Surface can be find out through following methods.

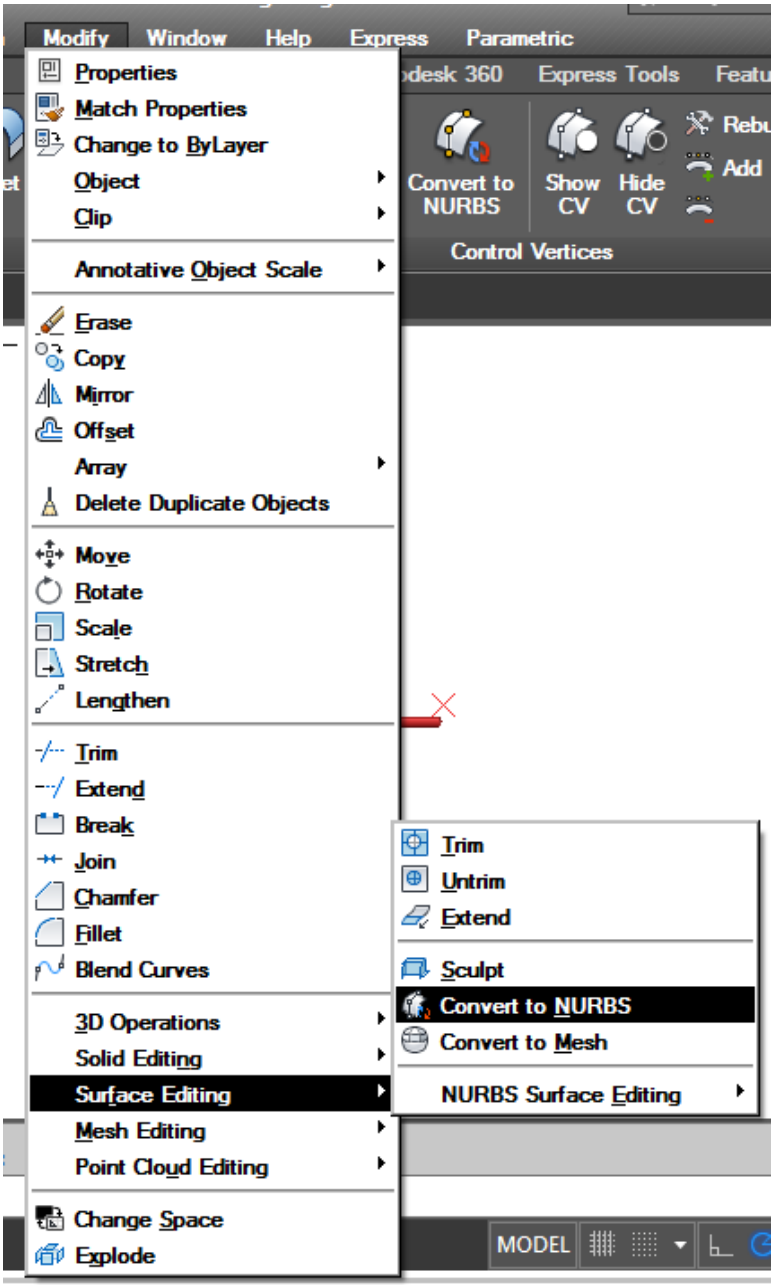
Through Ribbon

This figure describes the location to convert the NURBS Command of 3D Surface in Surface Tab.



Through Main Menu

Main Menu →Modify →Surface Editing →Convert to NURBS

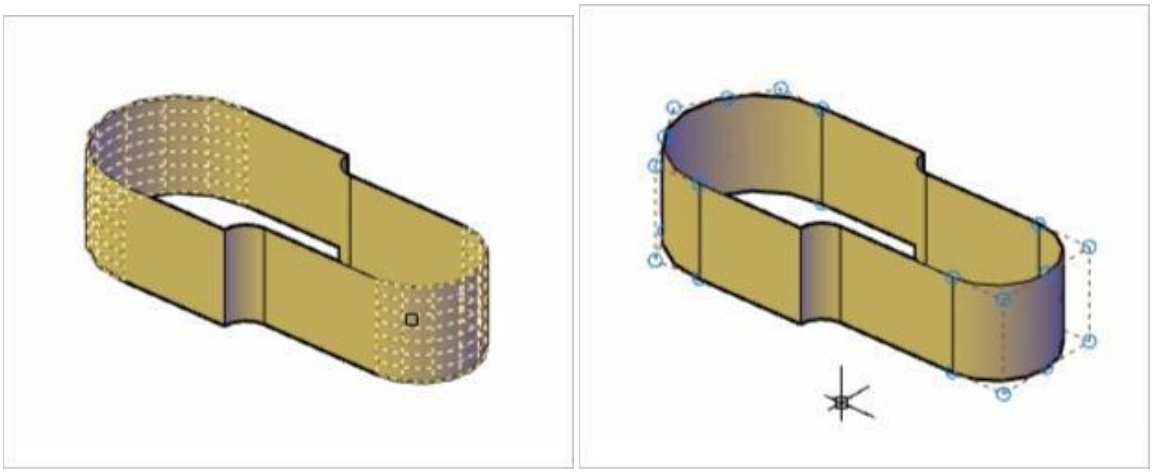


Through Short Cut Key

CONVTONURBS → enter

APPLICATION

Type CONVTONURBS at the Command Prompt → Select object to convert →enter



Convert to Surface

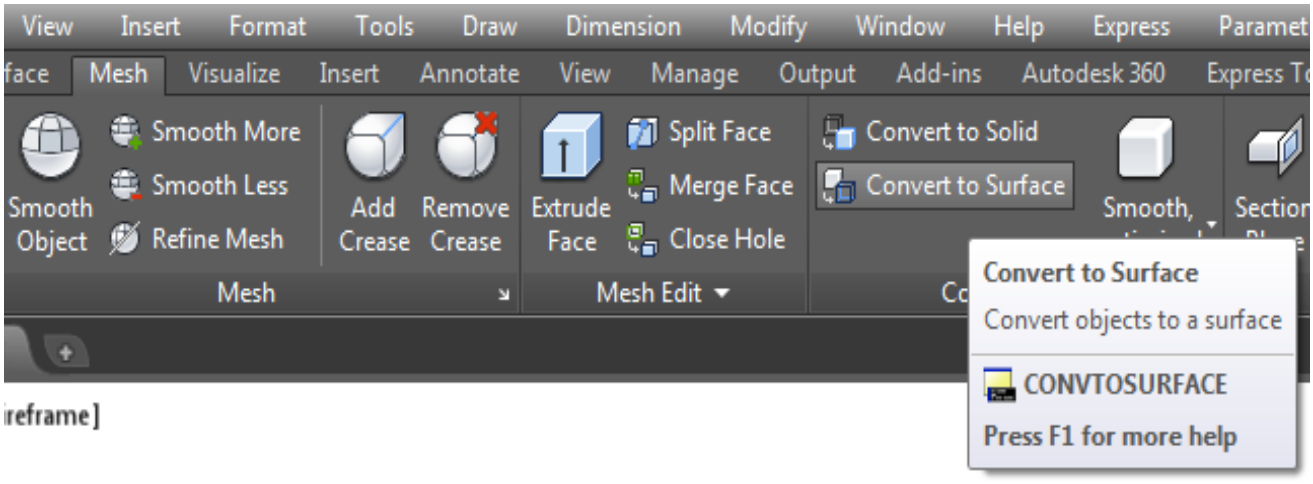
As you convert objects to surfaces, you can specify whether the resulting object is smooth or faceted.

When you convert a mesh, the smoothness and number of faces of the resulting surface are controlled by the SMOOTHMESHCONVERT system variable whereas the previous example shows a conversion to a smooth, optimized surface, the following example shows a conversion to a faceted surface in which the faces are not merged, or optimized.

Convert to Surface Command of 3D Surface can be find out through following methods.

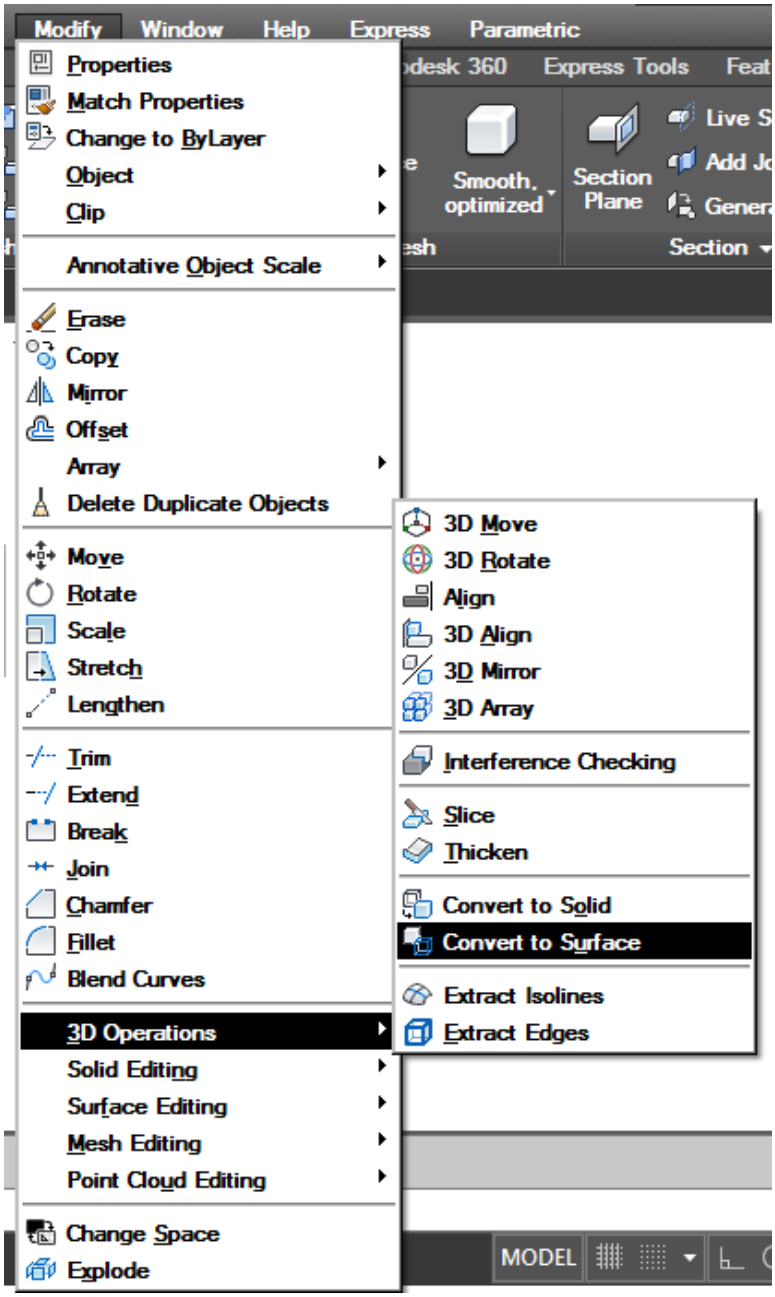
Through Ribbon

This figure describe the location of Convert to Surface Command of 3D Surface in Surface Tab



Through Main Menu

Main Menu →Modify →3D Operations →Convert to Surface

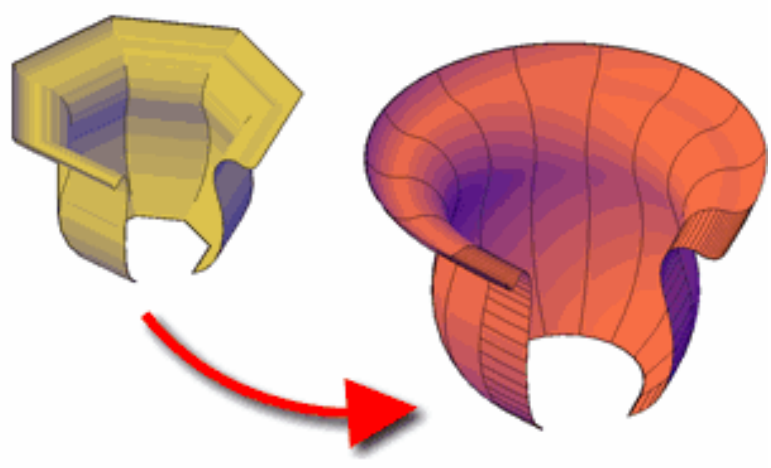


Through Short Cut Key

CONVTOSURFACE → enter

APPLICATION

Type CONVTOSURFACE at the Command Prompt →Select to object to convert →enter



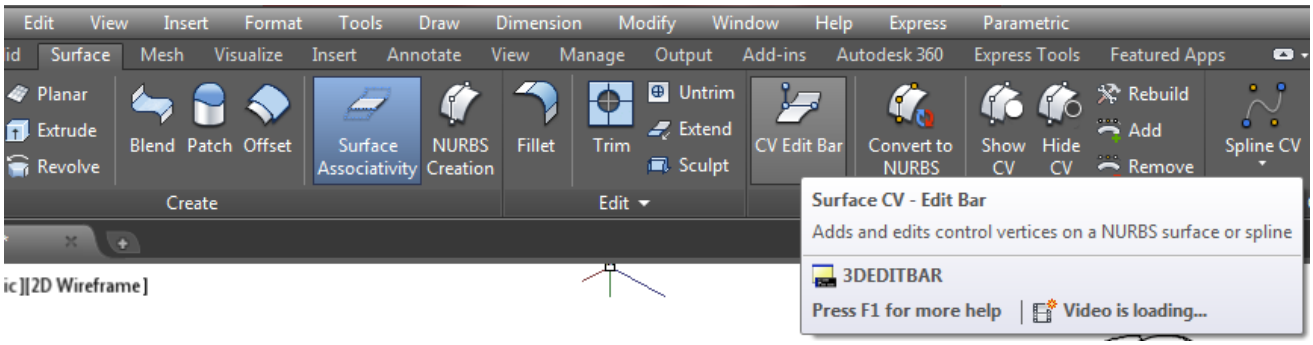
CV Edit BAR (3D Edit Bar)

Change the shape of NURBS surfaces and curves by using the 3D Edit Bar or by editing control vertices. Use the Control Vertices Edit Bar (3DEDITBAR) to drag and reshape surfaces. A second way to edit NURBS surfaces is to drag and edit the control vertices directly. Press and hold Shift to select multiple control vertices.

3D Edit Bar Command of 3D Surface can be find out through following methods

Through Ribbon

This figure describes the location of 3D Edit Bar Command of 3D Surface in Surface Tab.



Through Short Cut Key

3DEDITBAR → enter

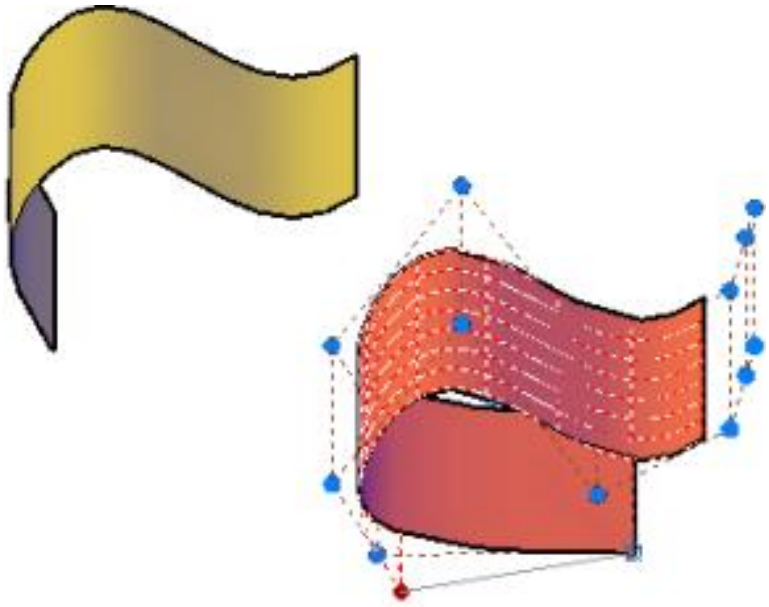
APPLICATION

Type 3DEDITBAR at the Command Prompt → Select to object Curve or Surface to convert →Show the control vertices→Drag the control vertices to reshape the curve or surface; you can also add or delete control vertices in both the U and V directions.

.

Remember

Use CVSHOW to display the control vertices for both NURBS surfaces and curves.



16. NURBS Vertex Control

Reconstruct NURBS surfaces and curves to a specified degree and number of control vertices. Editing a NURBS surface or curve can create discontinuity and wrinkles. Reconstruct the surface or curve by changing the degree and the number of control vertices. Rebuilding also allows you to delete the original geometry and for surfaces only to replace trimmed areas.

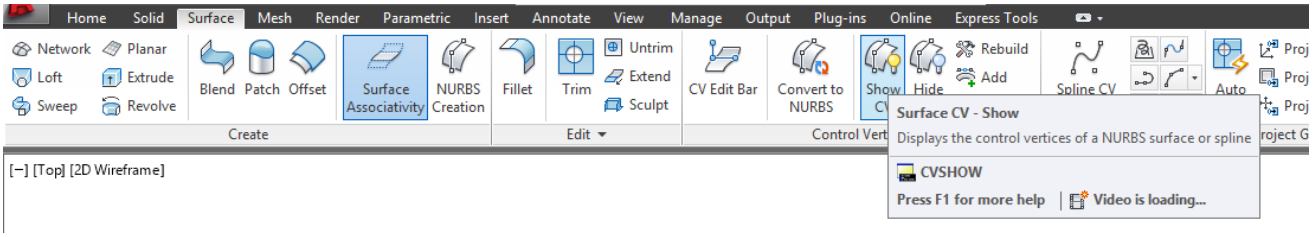
i. Surface CV – Show

Display the control vertices for specified NURBS surfaces or curves. Prompts you to select the NURBS surfaces and curves whose control vertices you want to display. Non-NURBS surfaces do not have control vertices. You can convert objects to NURBS surfaces with the CONVTONURBS command.

Surface CV Show Command of 3D Surface can be find out through following methods.

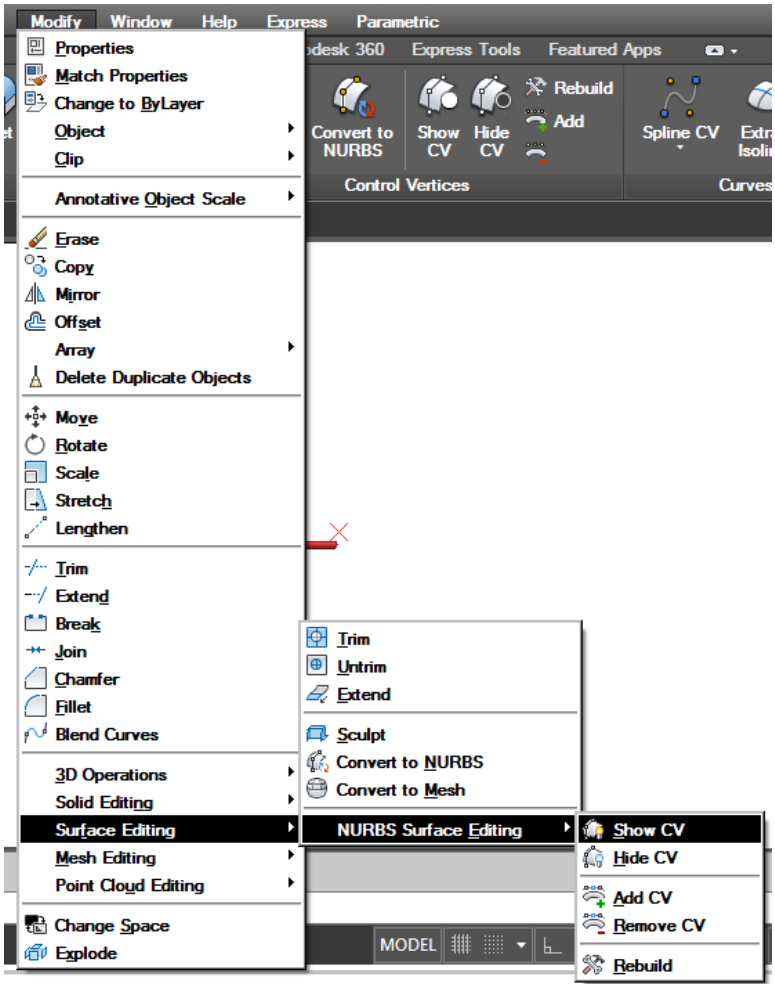
Through Ribbon

This figure describe the location of Surface CV Show Command of 3D Surface in Surface Tab



Through Main Menu

Main Menu →Modify →Surface Editing →NURBS Surface Editing →Show CV

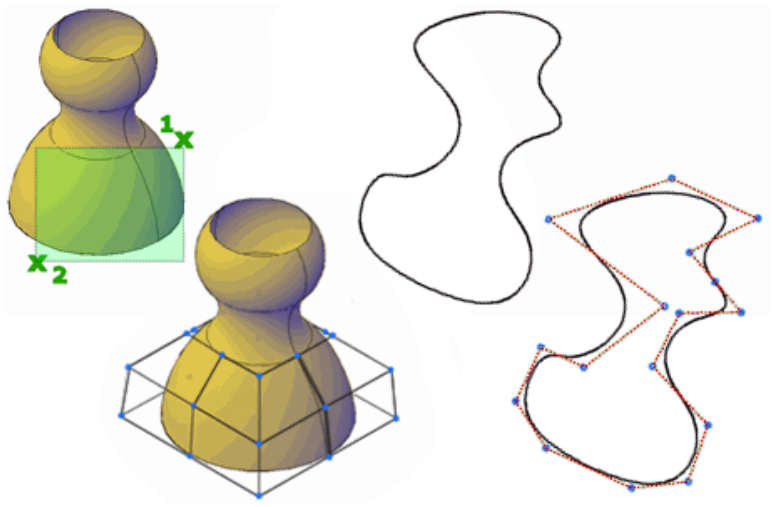


Through Short Cut Key

CVSHOW → enter

APPLICATION

Type CVSHOW at the Command Prompt →Select Curve or Surface →Show the control vertices

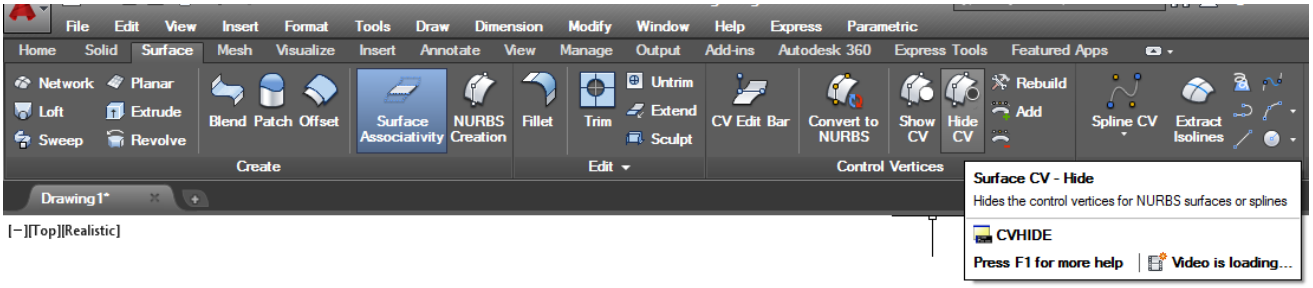


ii. Surface CV - Hide

Turn off the CV display for all NURBS objects. Surface CV Hide Command of 3D Surface can be find out through following methods.

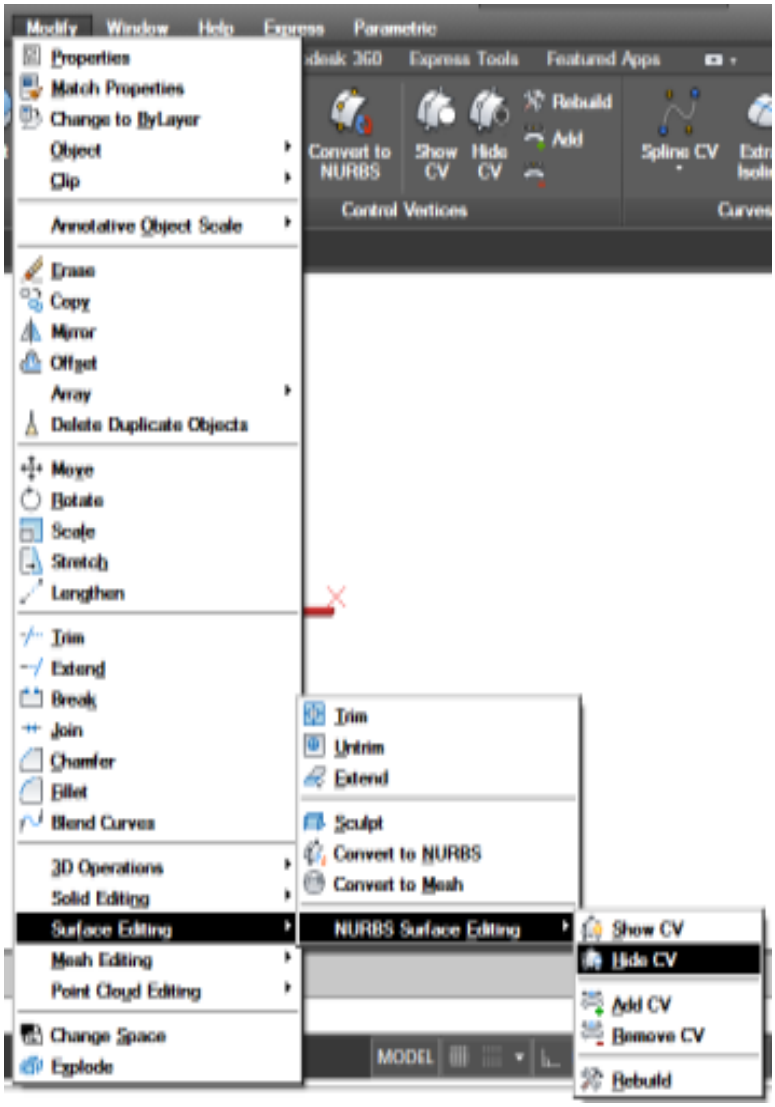
Through Ribbon

This figure describes the location of Surface CV Hide Command of 3D Surface in Surface Tab.



Through Main Menu

Main Menu →Modify →Surface Editing → NURBS Surface Editing →Hide CV

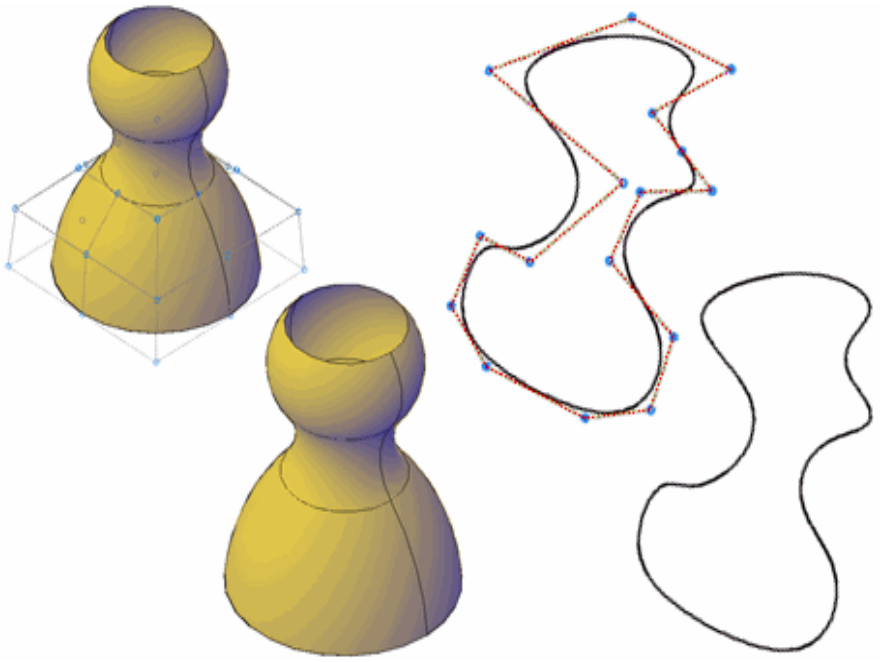


Through Short Cut Key

CVHIDE → enter

APPLICATION

Type CVHIDE at the Command Prompt → Select to Curve or Surface →Hide the control vertices



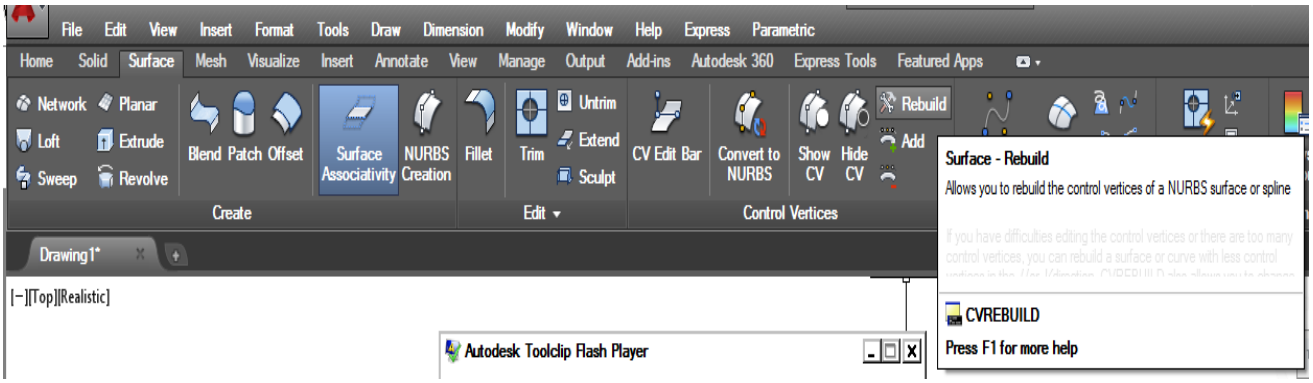
Surface CV Rebuild

Rebuild the shape of NURBS surfaces and curves. If you have difficulties editing the control vertices or there are too many control vertices, you can rebuild a surface or curve with less control vertices in the U or V direction. CVREBUILD also allows you to change the degree of the surface or curve.

Surface CV Rebuild Command of 3D Surface can be find out through following methods

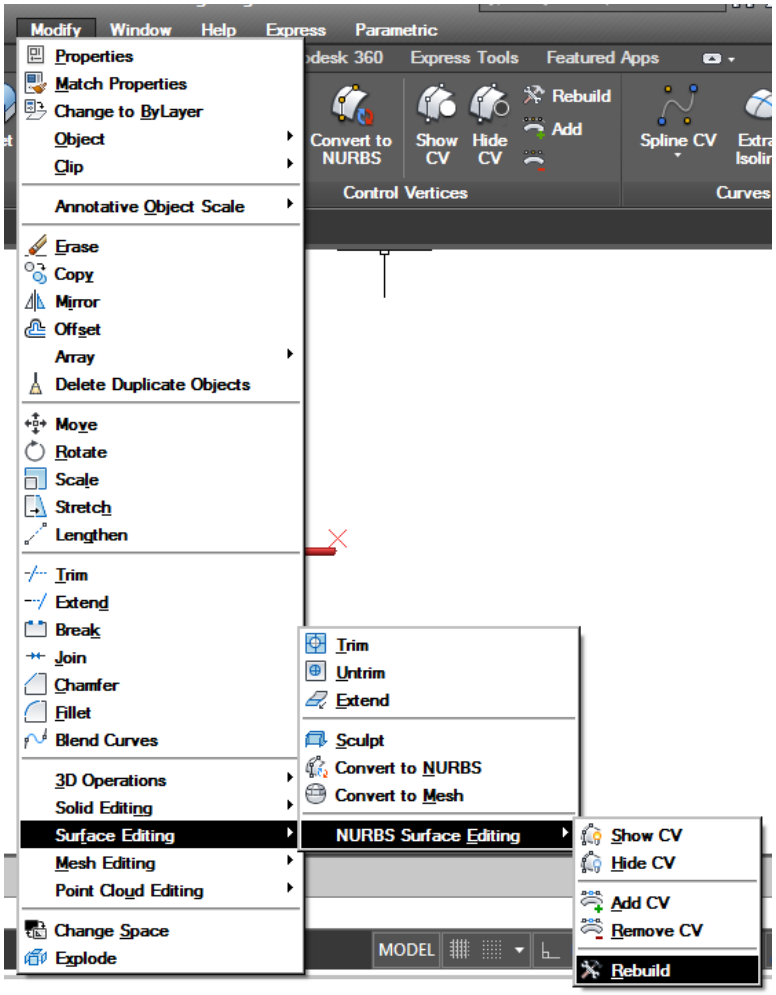
Through Ribbon

This figure describe the location of Surface CV Rebuild Command of 3D Surface in Surface Tab



Through Main Menu

Main Menu →Modify →Surface Editing → NURBS Surface Editing →Rebulid

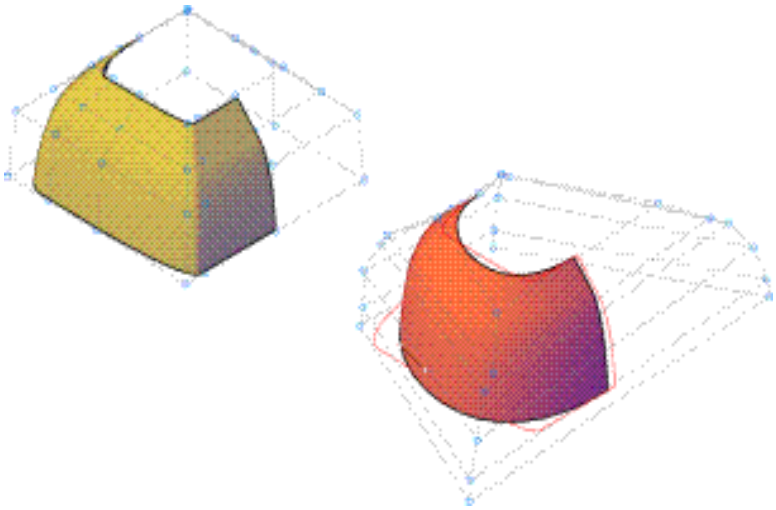


Through Short Cut Key

CVREBUILD → enter

APPLICATION

Type CVREBUILD at the Command Prompt →Select to Curve or Surface →Show the CV rebuild Dialog box



Note:

- If you select a **surface,** the Rebuild Surface dialog box is displayed.
- If you select a **spline**, the Rebuild Curve dialog box is displayed.

Note:
REBUILD U/V
Sets the number of grid lines in the U/V direction when rebuilding a NURBS Surface.
Valid values are 2 to 32767 to increase or decrease the number of grid lines in the U/V direction used to rebuild the NURBS surface.
Surfaces and Curves

Rebuild Surface dialog box

a) Control Vertices Count`

Specify the number of control vertices in the U and V directions.

b) In U Direction

Specify the number of control vertices in the U direction.

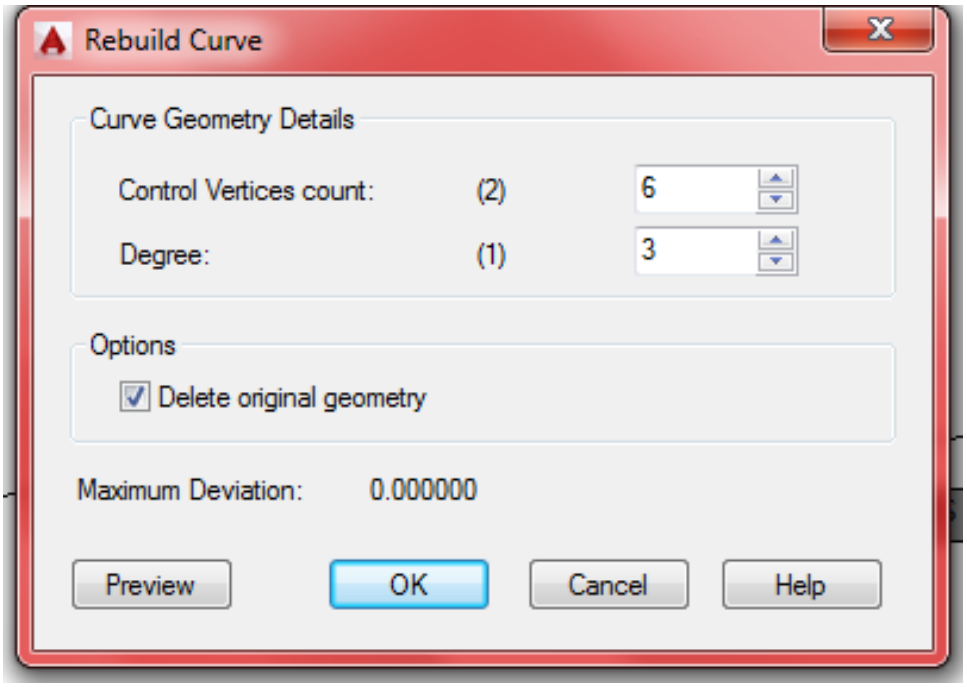
c) In V Direction

Specify the number of control vertices in the V direction.

d) Degree

Specify the number of control vertices available per span. The higher number, more complex the surface.

Rebuild Curve dialog box



a) Control Vertices Count

Specify the number of control vertices.

b) Degree

Specify the degree of the NURBS curve.

c) Options

Specify the build options.

d) Delete input Geometry

Specify whether the original curve is retained along with the rebuilt curve.

e) Maximum Deviation

Display the maximum deviation between the original curve and the new one.

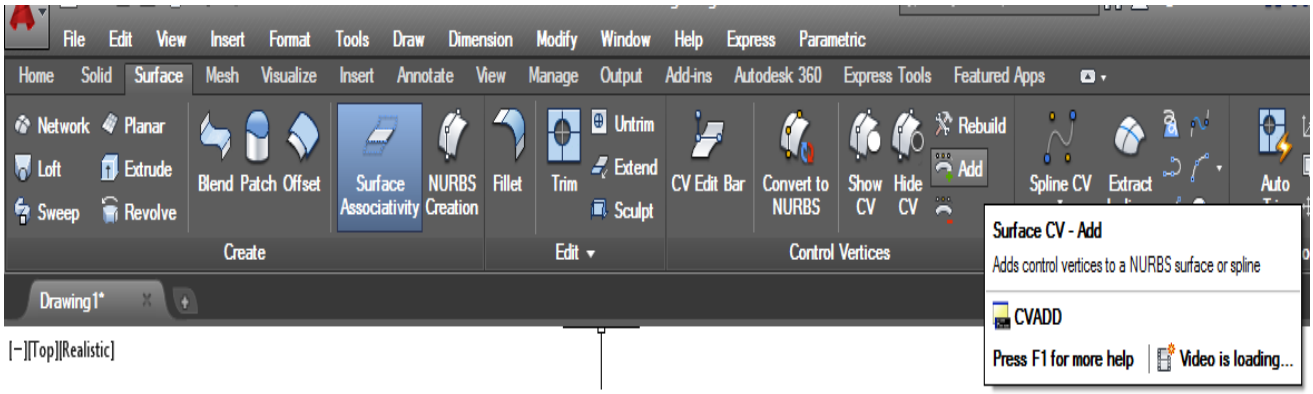
iii. Surface CV Add

Adds control vertices of NURBS surfaces or spline. Adds control vertices in the U or V direction, or adds points directly on a surface or spline. This illustration shows a row of control vertices added in the V direction.

Surface CV Add Command of 3D Surface can be find out through following methods

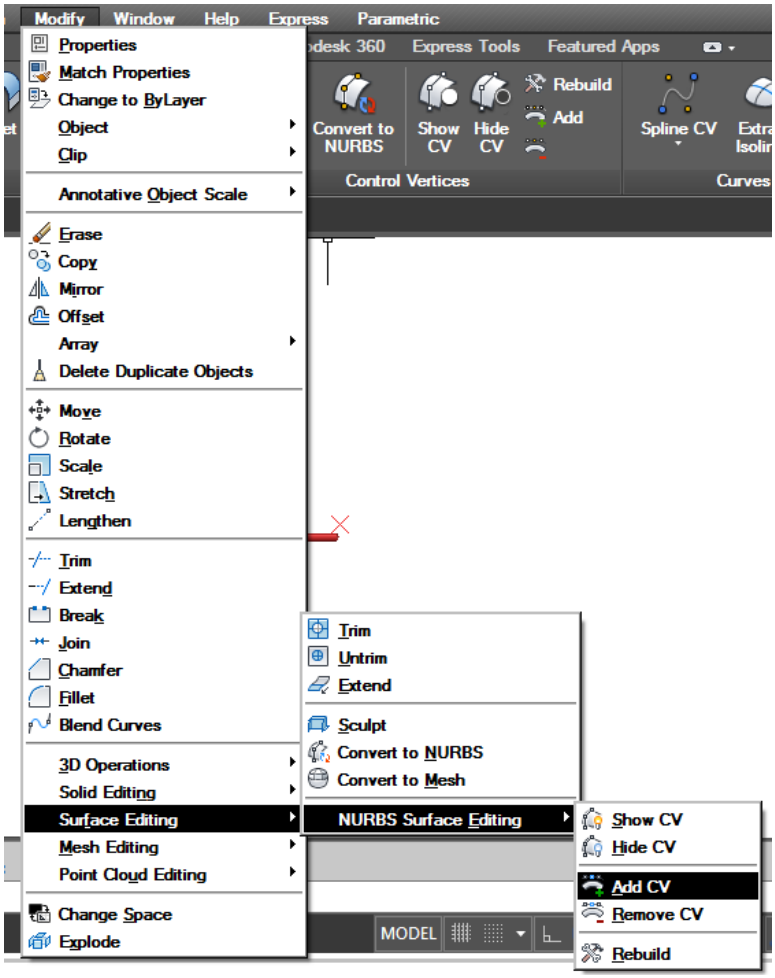
Through Ribbon

This figure describes the location of Surface CV Add Command of 3D Surface in Surface Tab.



Through Main Menu

Main Menu →Modify →Surface Editing → NURBS Surface Editing →Add CV

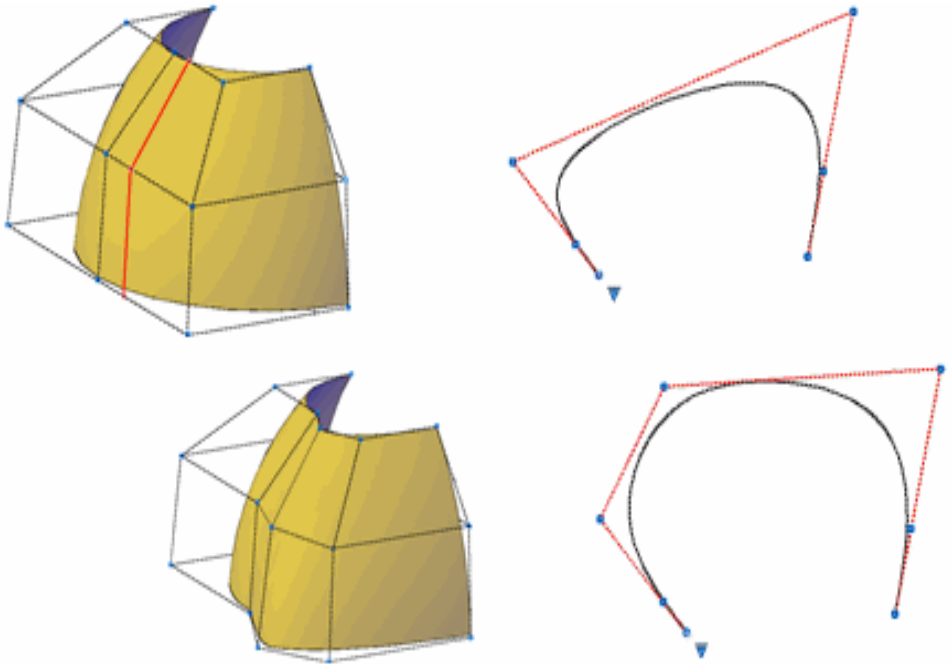


Through Short Cut Key

CVADD → enter

APPLICATION

Type CVADD at the Command Prompt →Select to Curve or Surface →Show the control vertices on object→select vertices to add CV →enter



iv. Surface CV Remove

Remove control vertices of NURBS surfaces or spline. Removes control vertices in the U or V direction. This illustration shows a row of control vertices deleted in the V direction.

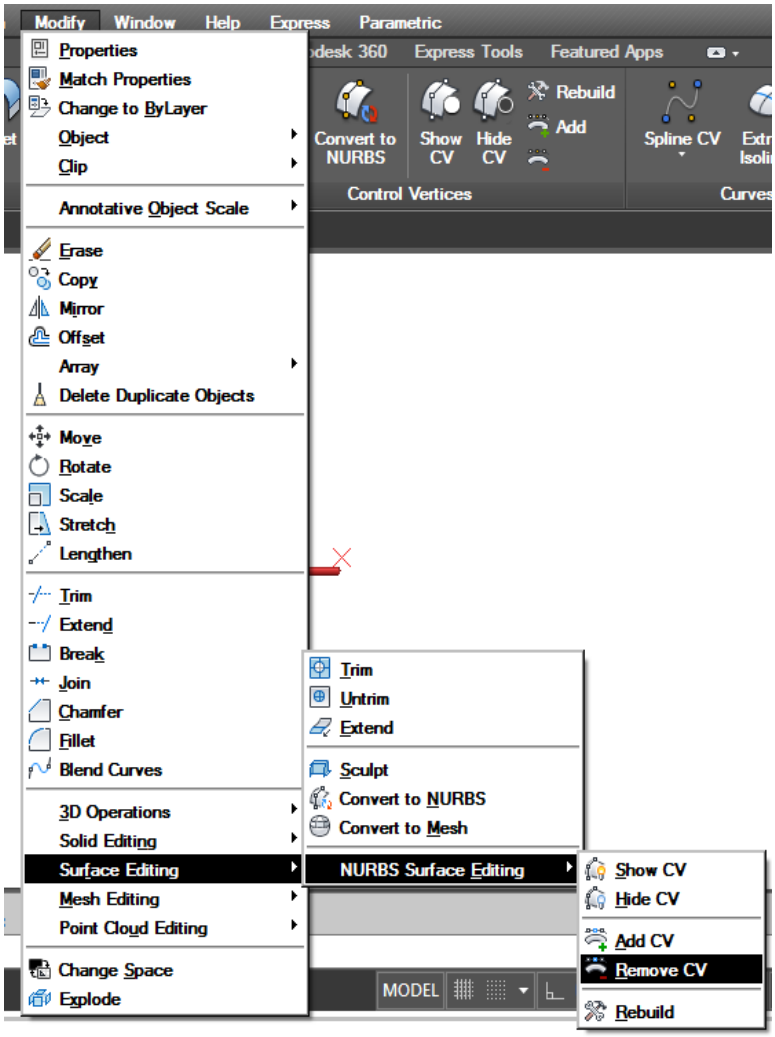
Surface CV Remove Command of 3D Surface can be find out through following methods

Through Ribbon

This figure describe the location of Surface CV Remove Command of 3D Surface in Surface Tab

Through Main Menu

Main Menu →Modify →Surface Editing → NURBS Surface Editing →Remove CV

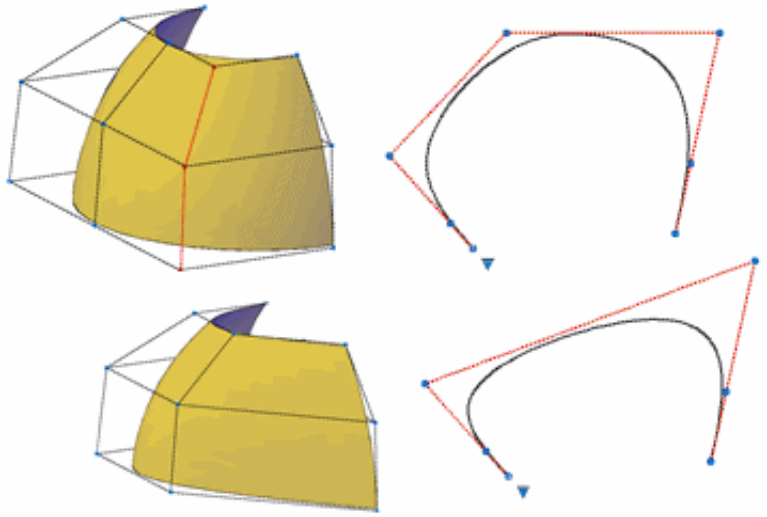


Through Short Cut Key

CVREMOVE → enter

APPLICATION

Type CVREMOVE at the Command Prompt →Select to Curve or Surface →Show the control vertices on object→select vertices to Remove CV →enter



Remember

The minimum number of control vertices that you can have on a surface or curve is two in any direction. If you try to remove more, an error displays.

17. Surface Analysis

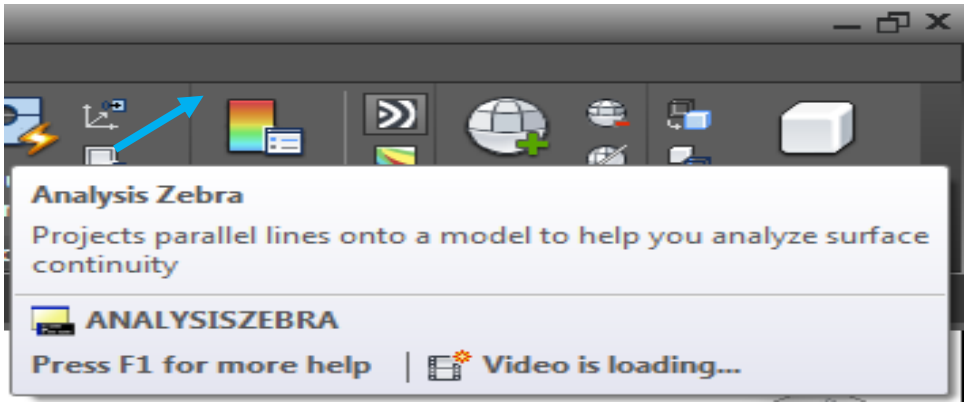
i. Analysis Zebra

Projects parallel lines onto a model to help you analyze surface continuity. The way the stripes line up where two surfaces meet helps analyze the tangency and curvature of the intersection.

Zebra Surface Analysis Command of 3D Surface can be find out through following methods.

Through Ribbon

This figure describes the location of Zebra Surface Analysis Command of 3D Surface in Surface Tab.



Through Short Cut Key

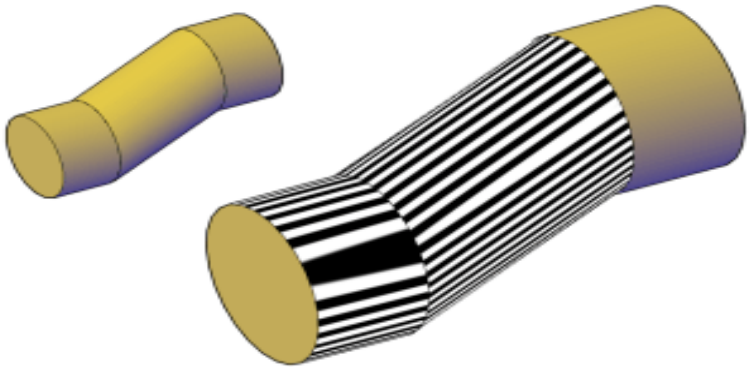
ANALYSISZEBRA → enter

APPLICATION

Type ANALYSISZEBRA at the Command Prompt →Select to Surface to apply zebra analysis →enter

Remember

Analysis tools only work in the 3D visual styles; they will not work in 2D



Zebra Stripes

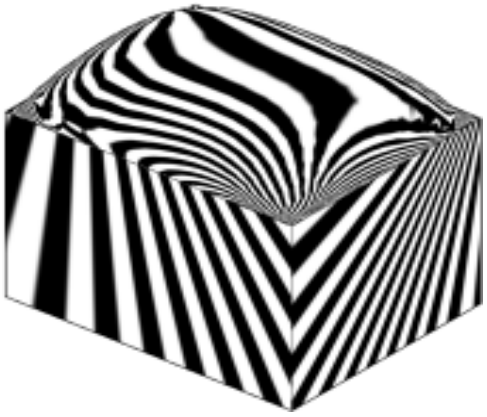
In the seam where two surfaces meet, the way that the zebra stripes align and curve tells you a lot about the smoothness of the join.

G0 Position.

The position of the surface edges is collocated; they touch. But the tangency and curvature do not match. The zebra stripes do not lineup.

G1 Tangency.

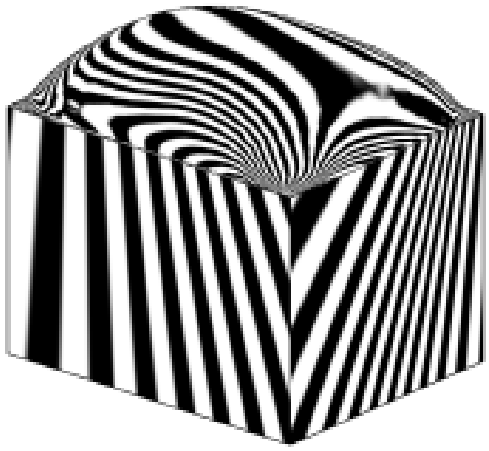
The position and tangency of surfaces is the same. This indicates G1 (G0 + G1 or position + tangency). The zebra stripes line up, but they veer away from one another at sharp curves.



G2 Curvature.

The position, tangency, and curvature of the surface edges are the same. This indicates G2 (G0 + G1 + G2 or position + tangency + curvature).

The stripes line up, but they do not veer away from each other at sharp curves (because they share the same curvature). This distinction is subtle and a little harder to discern from G1 continuity.



Analysis Curvature

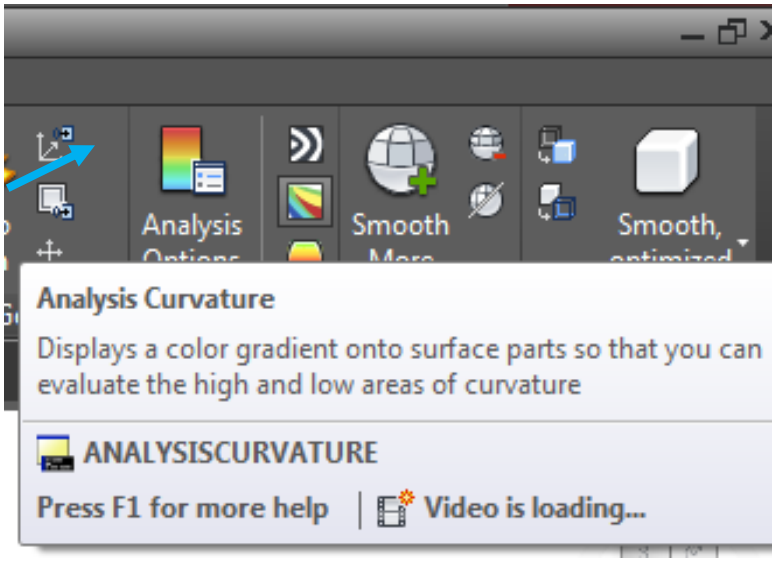
Display a color gradient onto surfaces to evaluate areas of high, low and Gaussian curvature. The color gradient allows you to visualize Gaussian, minimum, maximum, and mean U and V surface curvature. Maximum curvature and a positive Gaussian value display as red and minimum curvature and a negative Gaussian value display as blue. Positive Gaussian curvature means the surface is shaped like a bowl. Negative Gaussian curvature means the surface is shaped like a saddle. Mean curvature and a zero Gaussian value means the surface are flat in at least one direction (planes, cylinders, and cones have zero Gaussian curvature).

The curvature analysis tool ensures that your model stays within a specified range. You can analyze the following on a point on a surface or curve:

Analysis Curvature Command of 3D Surface can be find out through following methods.

Through Ribbon

This figure describe the location of Analysis Curvature Command of 3D Surface in Surface Tab

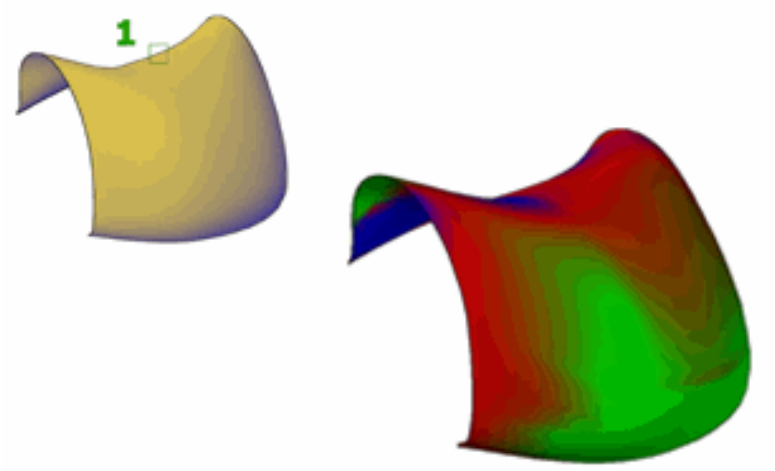


Through Short Cut Key

ANALSISCURVATURE → enter

APPLICATION

Type ANALYSISCURVATURE at the Command Prompt →Select to Surface to apply curvature analysis →enter



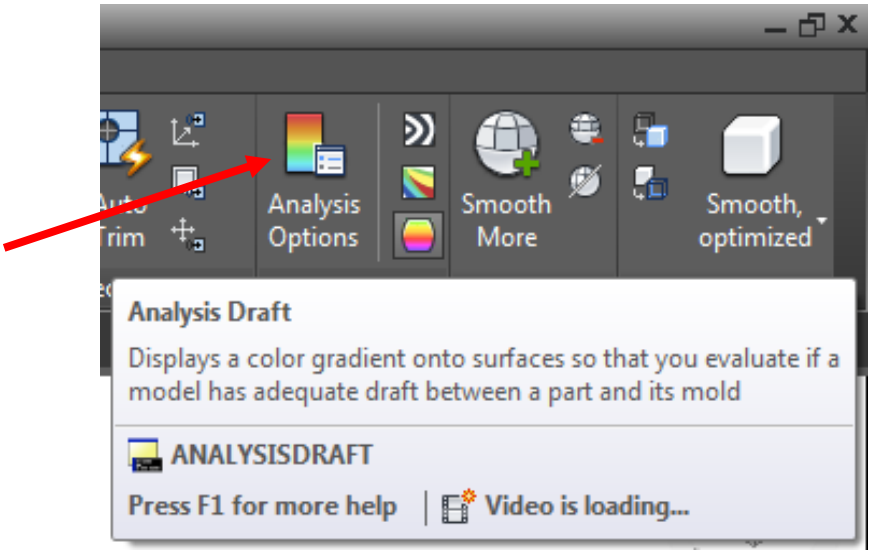
f) Analysis Draft

Display a color gradient onto a surface so you can evaluate if a model has adequate draft between a part and its mold. If you are creating shapes or parts that need to be molded, the draft analysis tool evaluates if there is adequate draft between a part and its mold (based on the pull direction).

Analysis Draft Command of 3D Surface can be find out through following methods

Through Ribbon

This figure describes the location of Analysis Draft Command of 3D Surface in Surface Tab.



Through Short Cut Key

ANALYSISDRAFT → enter

APPLICATION

Type ANALYSISDRAFT at the Command Prompt →Select to Surface or mold to apply draft analysis →enter

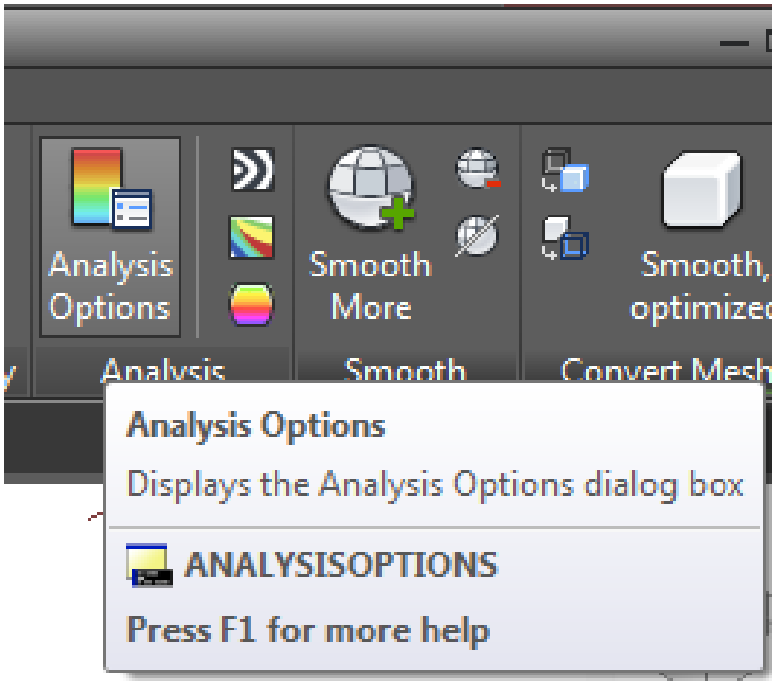
g) Analysis Options

Set the display options for zebra, curvature, and draft analysis.

Analysis Option for 3D Surface can be find out only through ribbon

Through Ribbon

This figure describes the location of Analysis Option for 3D Surface in Surface Tab.



Through Short Cut Key

ANALYSISOPTIONS → enter

APPLICATION

Type ANALYSISOPTIONS at the Command Prompt →Show the Analysis Dialog Box

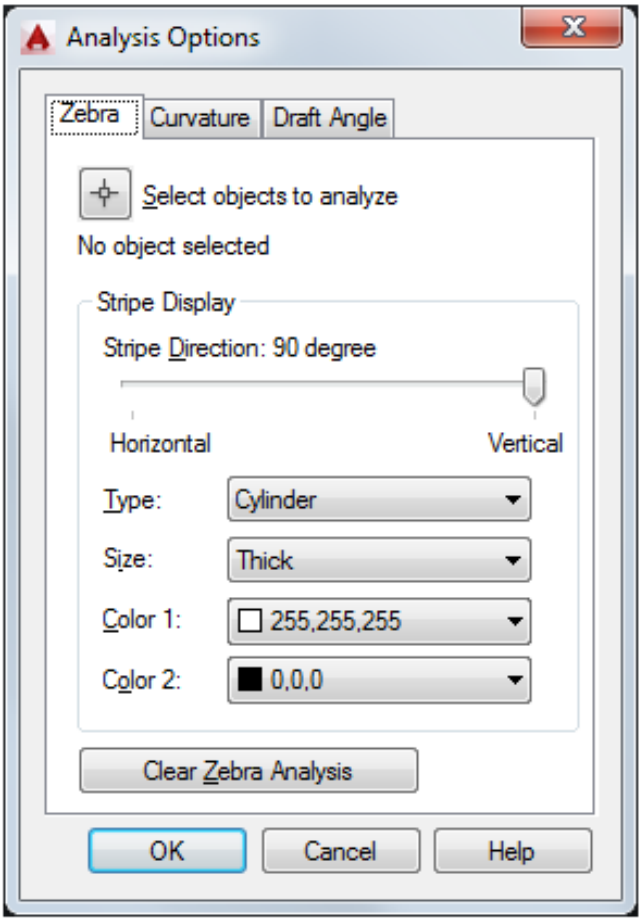
Show the Analysis Dialog Box

Set the display Dialog Box options for zebra, curvature, and draft angle analysis.

1) **Zebra Tab**
2) **Curvature Tab**
3) **Draft angle Tab**

1) <u>Zebra Tab</u>

Change the display settings for the continuity analysis of solid and surface objects.



i. Select Objects to Analyze

Prompt you to select the surface objects to analyze. When you are done selecting objects, press Enter to return to the dialog box

.

ii. Stripe Display

a) Stripe Direction
Specify the analysis display angle.
(VSAZEBRADIRECTION system variable)

b) Type
Set the analysis display type to Chrome Ball or Cylinder. (VSAZEBRATYPE system variable)

c) Size
Set the width of the zebra stripes.
(VSAZEBRASIZE system variable)

d) Color 1
Set the first color for zebra stripes.
(VSAZEBRACOLOR1 system variable)

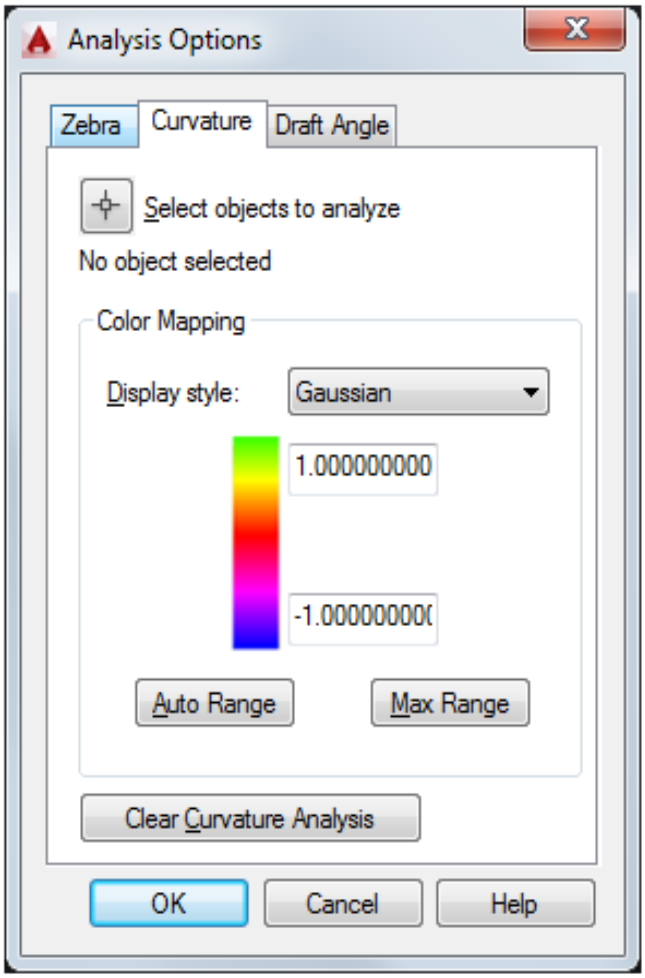
e) Color 2
Set the second color for zebra stripes.
(VSAZEBRACOLOR2 system variable)

iii. Clear Zebra Analysis

Removes the zebra display from all objects in the current drawing.

2) Curvature Tab

Change the display settings for the curvature analysis of solid and surface objects.



i. Select Objects to Analyze

Prompt you to select the surface objects to analyze. When you have done selecting objects, press Enter to return to the dialog box.

ii. Color Mapping

a) Display Style

Specify the display for color mapping.
(VSACURVATURETYPE system variable)

b) Auto Range

Calculate the curvature range so that 80% of the values are within the high and low range.
(VSACURVATUREHIGH and VSACURVATURELOW system variables)

c) Max Range

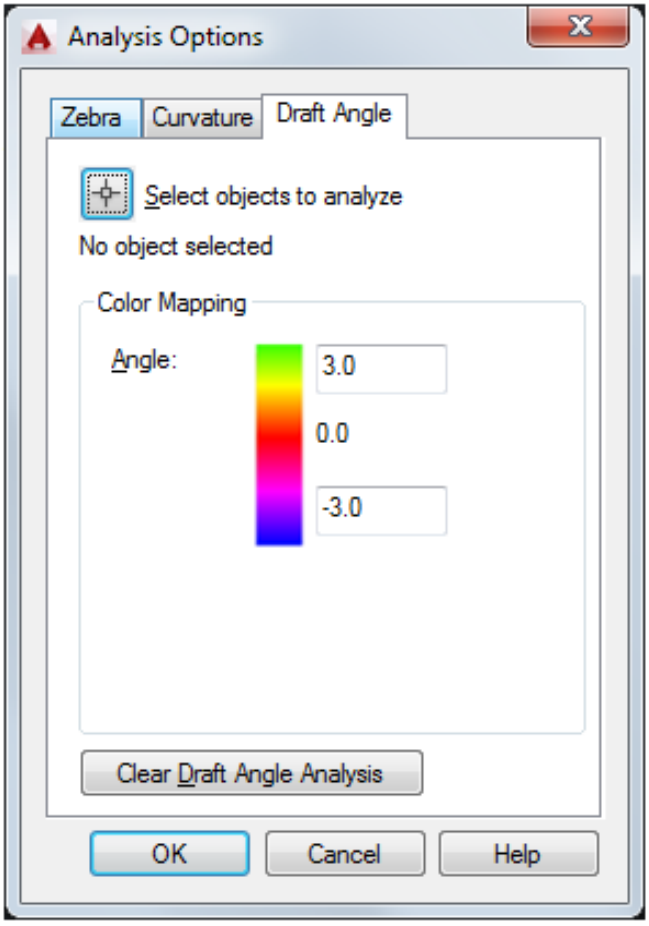
Calculate the maximum and minimum ranges of all objects selected for curvature analysis.
(VSACURVATUREHIGH and VSACURVATURELOW system variables)

iii. Clear Curvature Analysis

Remove the curvature analysis display from all objects in the current drawing.

3) Draft Angle Tab

Change the display settings for the draft analysis of solid and surface objects.



i. Select Objects to Analyze

Prompt you to select the surface objects to analyze. When you are done selecting objects, press Enter to return to the dialog box.

ii. Color Mapping

Maps green to the highest draft angle, red to the medium draft angle, and blue to the lowest draft angle

a) Angle

Set the value for the high and low draft angles. The draft angle is the angle in degrees between the surface normal and the UCS plane

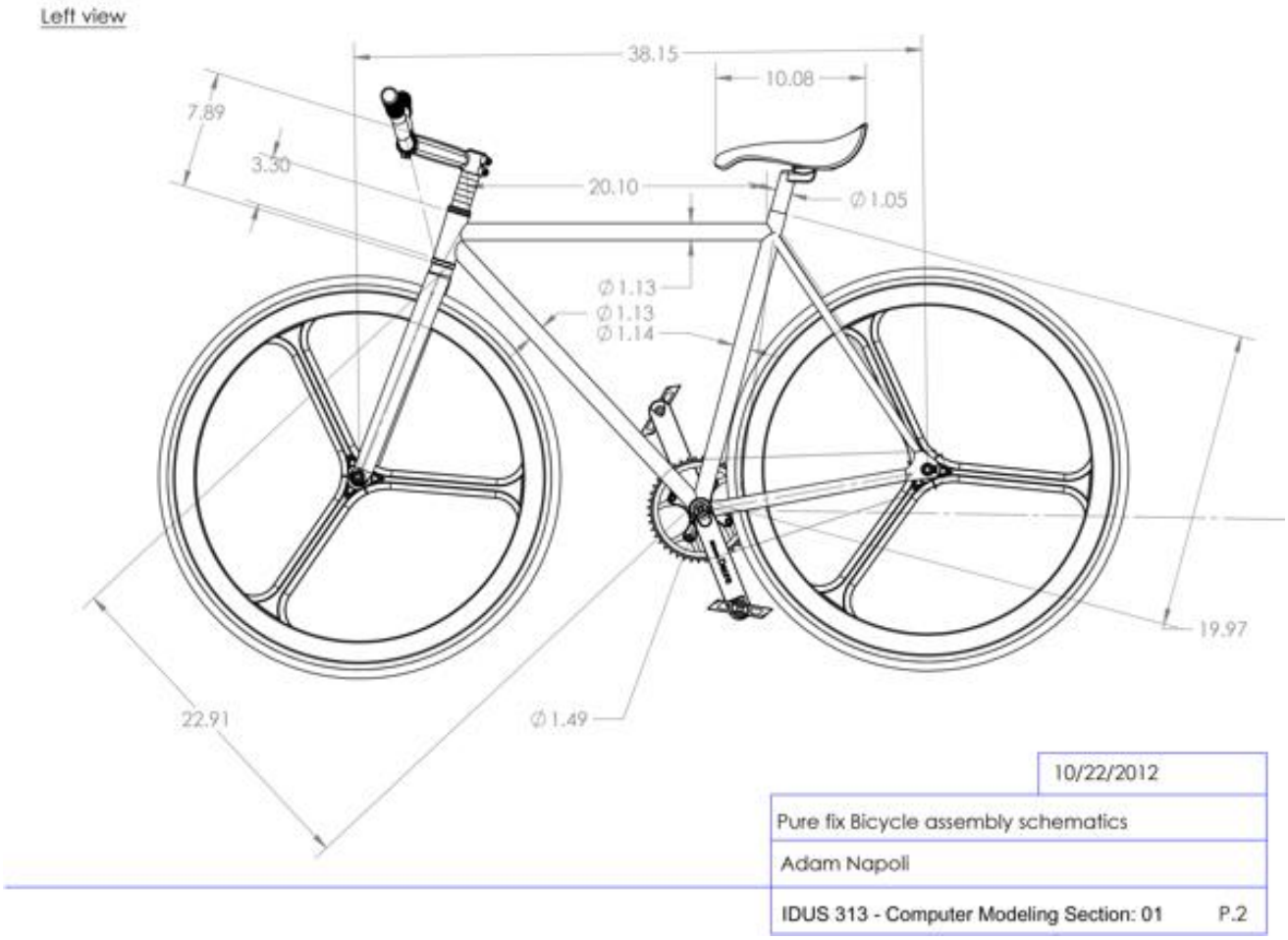
iii. Clear Draft Angle Analysis

Remove the curvature analysis display from all objects in the current drawing.

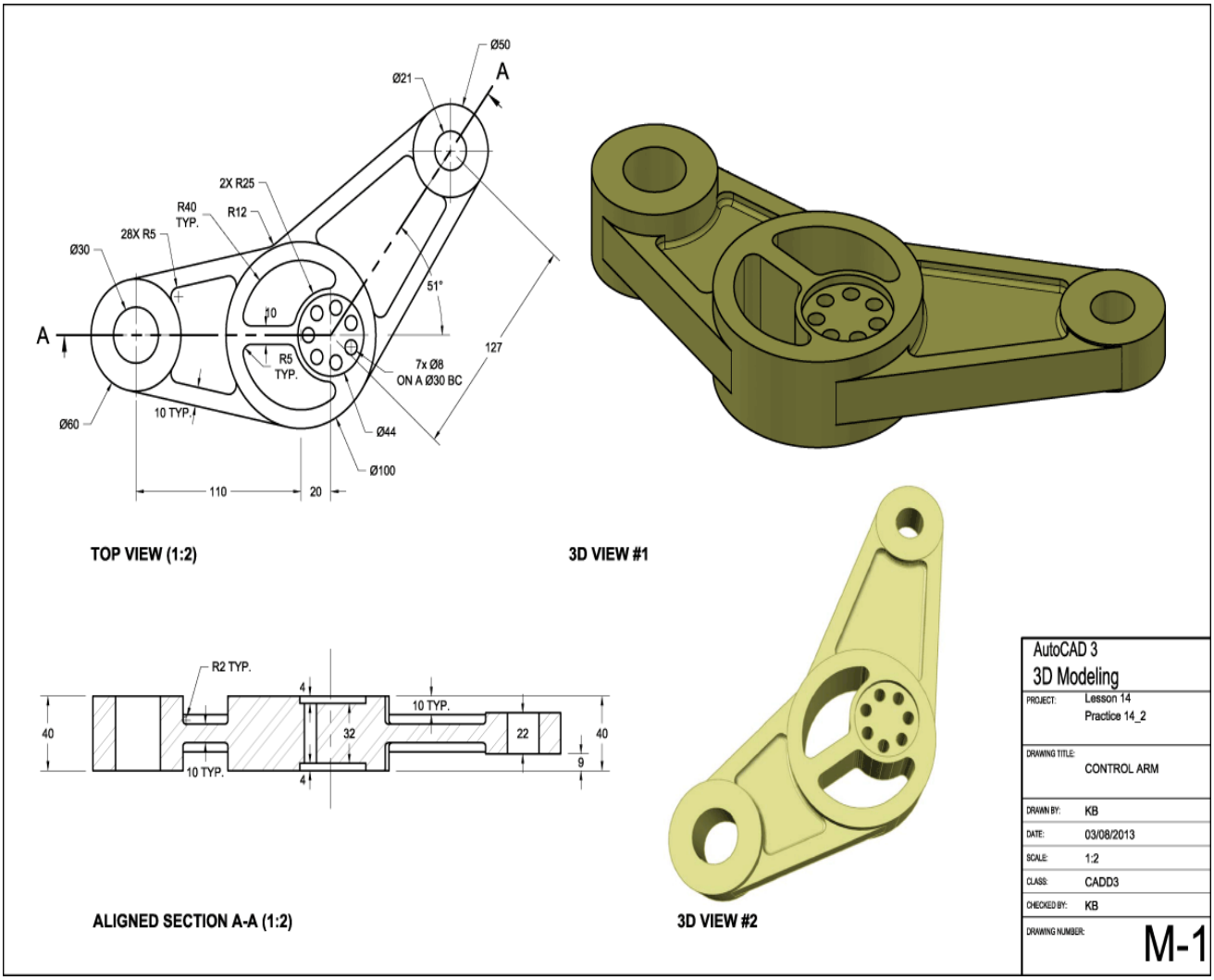
Learner's Activity

Draw the given projects according to requirements as mention by Using mesh and mesh edit commands

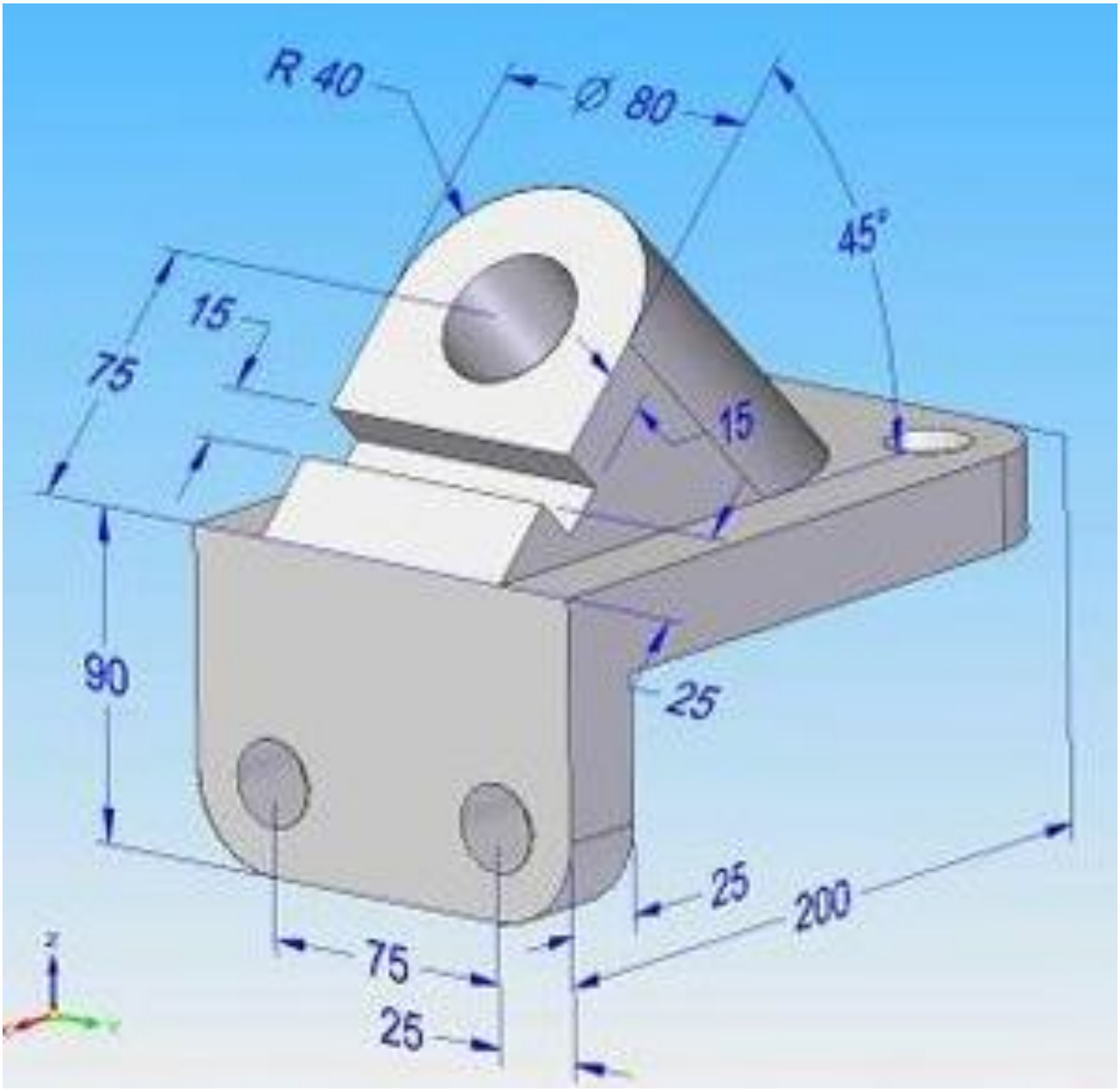
Project # 1



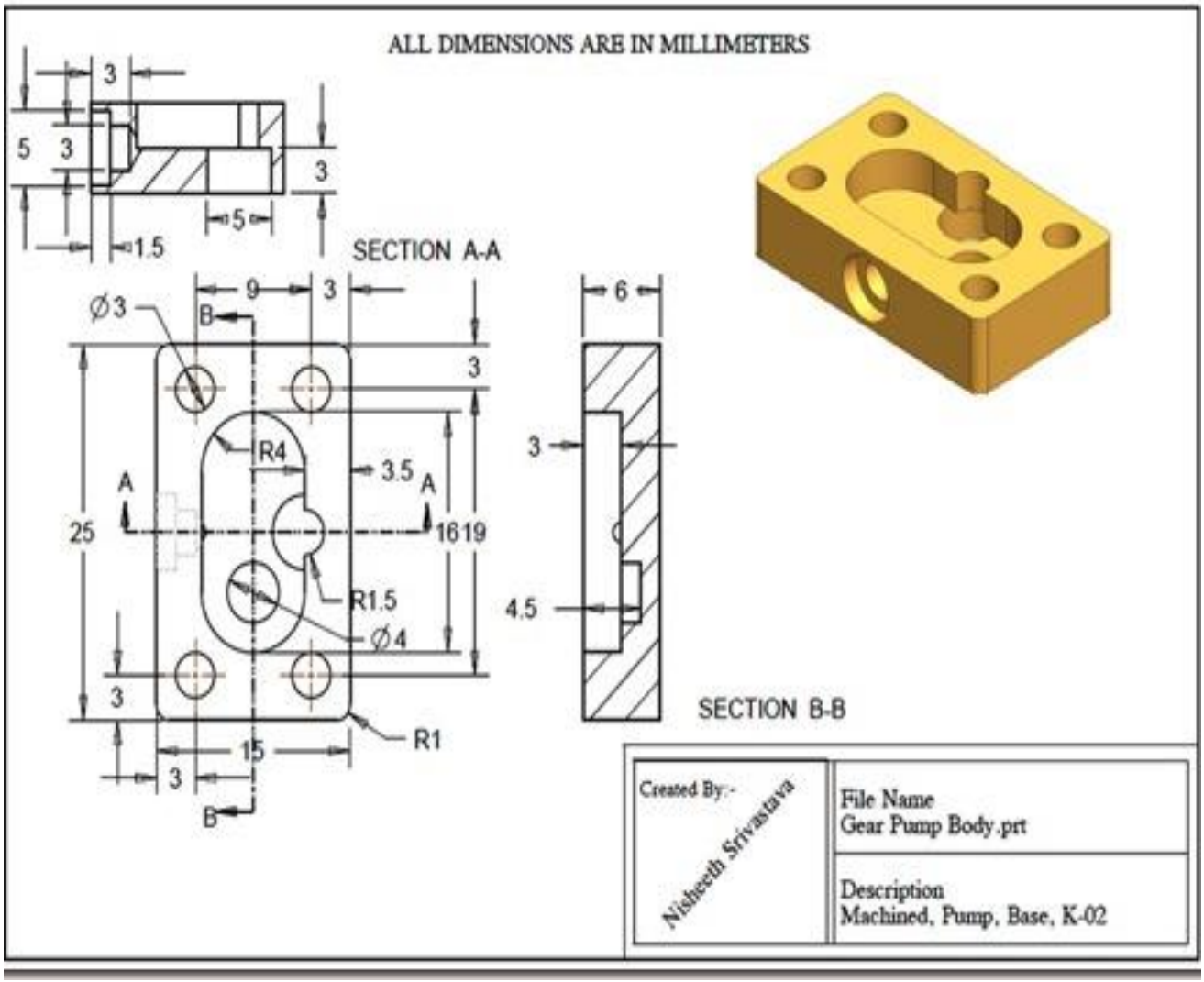
Project # 2


Ø50
A
Ø21
2X R25
R40
TYP.
R12
28X R5
Ø30
51°
10
A
R5
TYP.
7x Ø8
ON A Ø30 BC
127
Ø60
10 TYP.
Ø44
Ø100
110
20
TOP VIEW (1:2)
3D VIEW #1
R2 TYP.
4
10 TYP.
40
32
22
40
10 TYP.
9
4
ALIGNED SECTION A-A (1:2)
3D VIEW #2
AutoCAD 3
3D Modeling
PROJECT: Lesson 14
Practice 14_2
DRAWING TITLE: CONTROL ARM
DRAWN BY: KB
DATE: 03/08/2013
SCALE: 1:2
CLASS: CADD3
CHECKED BY: KB
DRAWING NUMBER: M-1

Project # 3



Project # 4


ALL DIMENSIONS ARE IN MILLIMETERS
SECTION A-A
SECTION B-B
Created By:-
Nisheeth Srivastava
File Name
Gear Pump Body.prt
Description
Machined, Pump, Base, K-02

Summary of Module

- The first part of this module described about fundamental mesh primitive command by which you can reshape mesh objects. You learnt how to perform various Mesh command such as mesh box used to create rectangular or cubical mesh box, Mesh cone used to draw cone, Mesh cylinder used to create cylinder of varying radius, Height, Mesh pyramid of inscribed / circumscribed perimeter, Mesh wedge and Mesh torus which was used to create ring shape surface of two radius value. You also learnt about the adjustment of the smoothness of mesh and how to add or remove mesh crease which helped you to sharpen or crease the edges of mesh objects. Editing meshes command such as Mesh extrude, Mesh split, Merge face, Mesh cap, helped you to edit mesh as you considered necessary.

- Whereas the second part of module described how to develop complex objects having 3D surfaces. You learnt to draw and edit complex surfaces, able to revolve a line, Arc, 2D/3D Polyline surface around specified axis, by using Surf tab's you learnt to control the density of mesh. You also learnt about tabulate surfaces, Ruled surface and adjustment of the edge surface. Furthermore you learnt to extrude surface in 2-axis and about loft Surface, their Settings , Sweep Surface, their alignment, Surface network, revolve Surface, Surface blend, Surface patches, Offset Surface, to edit surfaces by applying different commands; Such as surface fillet, Surface Trim/un trim, Surface extrude, Surface Sculpt and to draw NURBS Surfaces, Control NURBS vertex, finally you learnt to analysis Surface through different command; Analysis Zebra, Analysis curvature and Analysis draft.

Frequently Asked Questions (FAQs)

FAQ 1:	How many type of Primitive Meshes command are there?
Answer	There are seven type of Primitive Meshes command. 1) Mesh Box 2) Mesh Cone 3) Mesh Cylinder 4) Mesh Pyramid 5) Mesh Sphere 6) Mesh Wedge 7) Mesh Tours
FAQ 2:	How can you refine an object?
Answer	Refining a mesh object increases the number of editable faces, providing additional control over fine modeling details. To preserve program memory, you can refine specific faces instead of the entire object. Refining an object resets the smoothing level that was assigned to the object to 0 (zero). This level of smoothness becomes the new baseline for the object. That is, the smoothness level can no longer be decreased beyond that level. Refining a sub object does not reset the smoothing level.
FAQ 3:	What criteria should a guide curve meet?
Answer	Each guide curve must meet the following criteria • Intersects each cross section • Starts on the first cross section • Ends on the last cross section
FAQ 4:	How can you restore the trimmed area?
Answer	Replaces surface areas removed by the SURFTRIM command. If the trimmed edge is dependent on another surface edge that has also been trimmed, you may not be able to fully restore the trimmed area.
FAQ 5:	What is the loft Surface?
Answer	Create 3D solid or surface by specifying a series of cross sections. The cross sections define the shape of the resulting solid or surface. You must specify at least two cross sections. Loft profiles can be open or closed, planar or non-planar, and can also be edge sub objects. Use the mode option to select whether to create a surface or a solid

FAQ 6:	What can you do by applying surface sculpt command?
Answer	The SURFSCULPT command automatically combines and trims a collection of surfaces that enclose a watertight area to create a solid. The SURFSCULPT command also works with solid and mesh objects
FAQ 7:	What is NURBS?
Answer	Non-Uniform Rational B-Splines, are mathematical representations of 3-D geometry that can accurately describe any shape from a simple 2-D line, circle, arc, or curve to the most complex 3-D organic free-form surface or solid.
FAQ 8:	Describe method for the creation of NURBS surface?
Answer	There are Two Methods for Creating NURBS Surfaces i. SURFACEMODELINGMODE system variable. Use any of the surface creation commands while this system variable is set to 1. ii. CONVTONURBS command. Convert any existing procedural surfaces to NURBS surfaces with this command.
FAQ 9:	How can you rebuild the shape of NURBS Surface?
Answer	Rebuild the shape of NURBS surfaces and curves. If you have difficulties editing the control vertices or there are too many control vertices, you can rebuild a surface or curve with less control vertices in the U or V direction. CVREBUILD also allows you to change the degree of the surface or curve
FAQ 10:	How many type of Surface Analysis?
Answer	There are three type of Surface Analysis i. Zebra Analysis ii. Curvature Analysis iii. Draft Analysis

Test Yourself!

Please mark the correct one from the given options. You can check your answer with the Answer Key at the end of this module.

1. How many types of Primitive Meshes Command are there?

a. Four
b. Six
c. Five
d. Seven

2. TTR Stands for?

a. Torus Tangent Radius
b. Tangent Torus Radius
c. Tangent ,Tangent ,Radius
d. Torus Torus Radius

3. You can increase the smoothness level up to…

a. 3
b. 4
c. 0
d. 5

4. You can select Sub object with the help of…

a. Ctrl key
b. Ctrl + shift key
c. Shift key
d. None of all

5. How many types of Surface Analysis are there?

a. 3
b. 2
c. 4
d. 5

Answer Key

MCQ Number	Correct Answer
1	d
2	c
3	b
4	a
5	a

AUTOCAD

Module-8

LEARNING GUIDE

National Vocational Certificate Level 3

Version 1 - July 2013

Module 8: Develop Solids

Learning Outcomes

After completion of this learning module, you will be able to:

- Create Solids
- Edit Effectively 3D Objects
- Develop 3D Solid Composites

Learning Unit 1: Create Solids

Overview

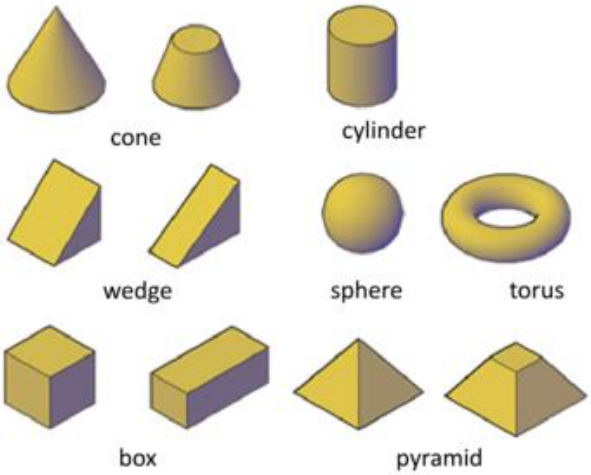
In this learning Unit, You will learn about the creating of 3D Solid object At the end of this unit, You will be able to create boxes, cones, cylinders spheres, wedges, pyramids and tore. You can draw more complex solids by joining to solids, Subtract one from other, you can control the size and rotation of the box, shapes of cone, convert an existing line, 2D Polylines, are or circle to a solid with rectangular profile. By using boundary command and polyline edit command, you can convert a line into polyline, Able to extrude an open / closed object to draw 3D Surface or solid. You would be able to apply revolve command in order to create a surface or solid, Sweep command for drawing solid by sweeping an open or close, planner or non-planner curve along an open or close path, left command to draw a solid by joining diff, cross sections lying along diff planes

1. Solid Primitive Commands

The basic solid forms, known as solid primitives, are the equivalent of the primitive forms for 3D solids.

You can create several basic 3D shapes, known as solid primitives: boxes, cones, cylinders, spheres, wedges, pyramids, and torus (donuts).

By combining primitive shapes, you can create more complex solids. For example, you can join two solids, subtract one from the other, or create a shape based on the intersection of their volumes.



Solid Primitives

Solid primitives can easily be drawn from the Home Tab & Solid Tab

1) **Box**
2) **Cone**
3) **Cylinder**
4) **Pyramid**

5) **Sphere**
6) **Wedge**
7) **Tours**

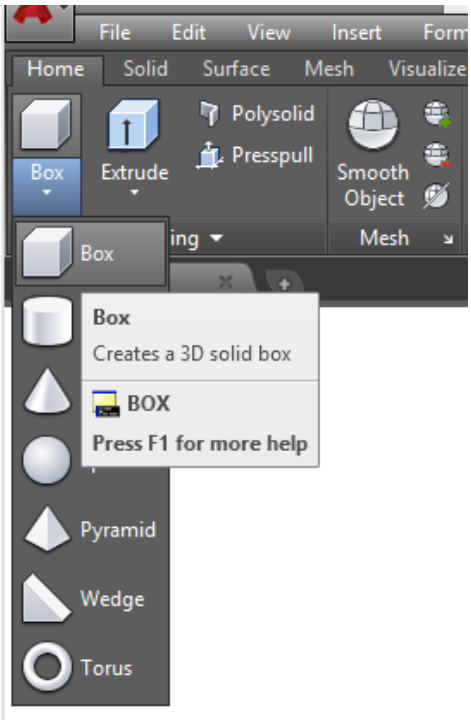
1) Box

The base of the box is always drawn parallel to the XY plane of the current UCS (work plane). The height of the box is specified in the Z-axis direction. You can enter both positive and negative values for the height.

Box command of 3D Solid can be find out through the following methods:

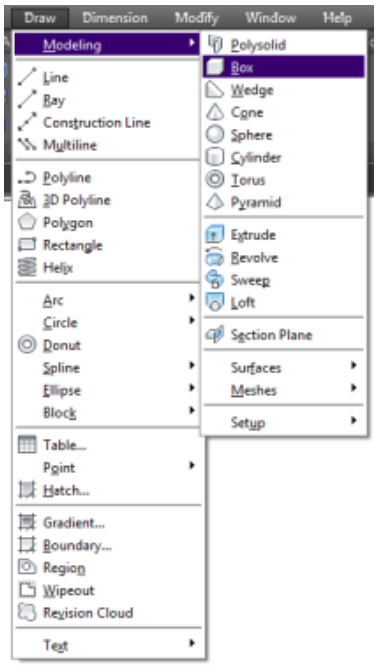
Through Ribbon

This figure describes the location of Box command of 3D Solid in Home Tab.



Through Main Menu

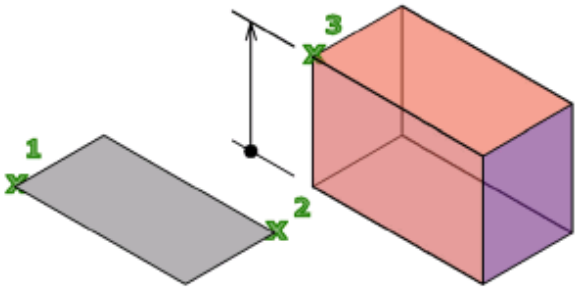
Main Menu →Draw →Modeling →Box



Through Short Cut Key →Box enter

Application

Type **Box** at the Command Prompt →Specify Start point→Put the Value of @ X and Y Co-ordinate→ Put the value of height →enter



Box Options

Use the following options to control the size and rotation of the boxes you create:

i. Cube

ii. Length

iii. Center Point

i. Cube

Use the Cube option of the BOX command to create a box with sides of equal length.

ii. **Length**
Create a box with length, width, and height values you specify. If you enter values, the length corresponds to the X axis, the width to the Y axis, and the height to the Z axis. If you pick a point to specify the length, you also specify the rotation in the XY plane.

iii. **Center point.**
Use the Center option to create a box using a specified center point.

2) Cone

Create a 3D solid with a circular or elliptical base that tapers symmetrically to a point or to a circular or elliptical planar face. You can control the smoothness of 3D curved solids, such as a cone, in a shaded or hidden visual style with the FACETRES system variable.

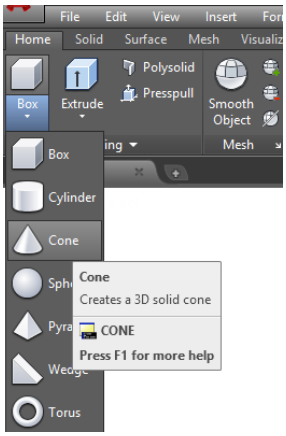
Use the Top Radius option to create a cone frustum.

Initially, the default base radius is not set to any value. During a drawing session, the default value for the base radius is always the previously entered base radius value for any solid primitive.

Cone command of 3D Solid can be find out through following methods:

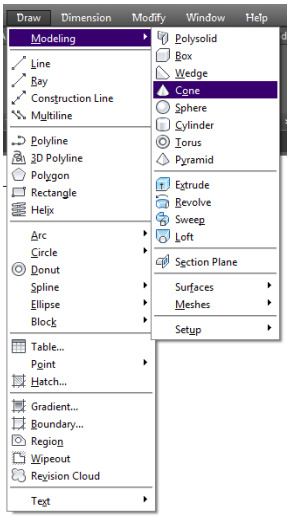
Through Ribbon

This figure describes the location of Cone command of 3D Solid in Home Tab.



Through Main Menu

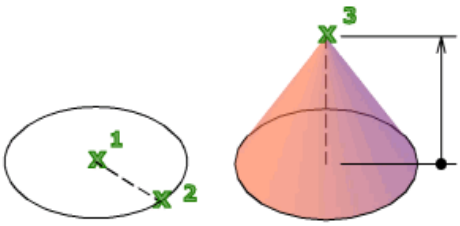
Main Menu →Draw →Modeling →Cone



Through Short Cut Key → Cone enter

Application

Type **Cone** at the Command Prompt → Specify center point of base→put the base radius value→Put the Value of Height→enter



Cone Option
Use the following options to control the size and rotation of the cones you create.

i. Center Point

ii. Axis Endpoint

iii. Frustum

iv. 3P

v. Elliptical

vi. TTR(Tangent, Tangent, Radius)

i. Center Point

When you specify the center point, the Cone is positioned so that its central axis is parallel to the Z axis of the current user coordinate system (UCS). Latitudinal lines are parallel to the XY plane.

i. **Radius** Defines the radius of the Cone.

ii. **Diameter** Defines the diameter of the Cone.

ii. Axis Endpoint

Use the Axis Endpoint option when you want to reorient the cone by placing the tip or axis endpoint anywhere in 3D space.

iii. Frustum

Use the Top Radius option to create a frustum of a cone, which tapers to an elliptical or planar face.

iv. 3P (Three Points)

The 3P (Three Points) option defines the size and plane of the base of the cone anywhere in 3D space.

v. Elliptical

Use the Elliptical option to create a cone base whose axes are of different lengths.

vi. TTR(Tangent, Tangent, Radius)

Use the TTR (Tangent, Tangent, and Radius) option to define points on two objects. Depending on the radius distance, the new cone is located as near as possible to the tangent points you specify. You can set up tangency with circles, arcs, lines, and some 3D objects. The tangency points are projected into the current UCS. The appearance of tangency is affected by the current level of smoothness.

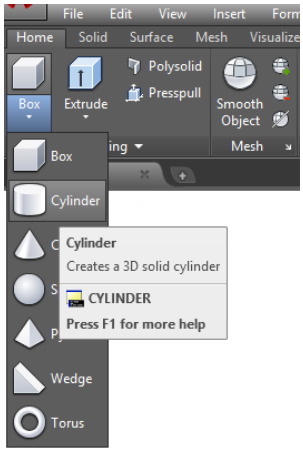
3) Cylinder

In the illustration, the cylinder was created using a center point (1), a point on the radius (2), and a point for the height (3). The base of the cylinder is always on a plane parallel with the work plane. You can control the smoothness of curved 3D solids, such as a cylinder, in a shaded or hidden visual style with the FACETRES system variable.

Cylinder command of 3D Solid can be find out through following methods

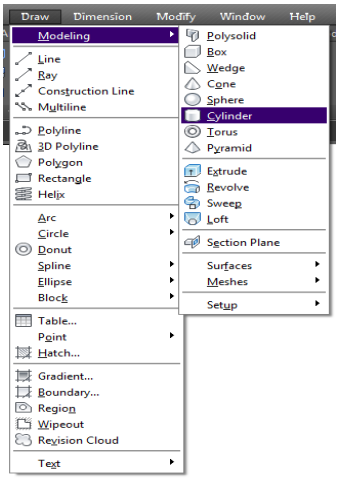
Through Ribbon

This figure describes the location of Cylinder command of 3D Solid in Home Tab.



Through Main Menu

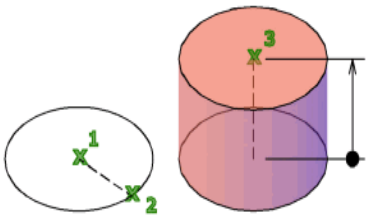
Main Menu →Draw →Modeling →Cylinder



Through Short Cut Key → Cylinder enter

Application

Type **Cyl** at the Command Prompt → Specify center point of base→put the base radius value→Put the Value of Height →enter



Remember

During a drawing session, the default value for the base radius is always as the previously entered base radius.

Cylinder Options

The Cylinder option provides several methods for determining the size and rotation of the cylinders you create

i. Axis Endpoint

ii. 3P(Three Point

iii. Elliptical

iv. TTR(Tangent, Tangent, Radius)

i. Axis Endpoint

Use the Axis Endpoint option to set the height and rotation of the cylinder. The center point of the top plane of the cylinder is the axis endpoint, which can be locate anywhere in 3D space.

ii. 3P (Three Points)

Use the 3P (Three Points) option to define the base of the cylinder. You can set three points anywhere in 3D space.

iii. Elliptical

Use the Elliptical option to create a cylinder base whose axes are of different lengths.

iv. TTR(Tangent, Tangent, Radius)

Use the TTR (Tangent, Tangent, and Radius) option to define points on two objects. Depending on the radius distance, the new cylinder is located as near as possible to the tangent points you specify. You can set up tangency with circles, arcs, lines, and some 3D objects. The tangency points are projected onto the current UCS.

Pyramid

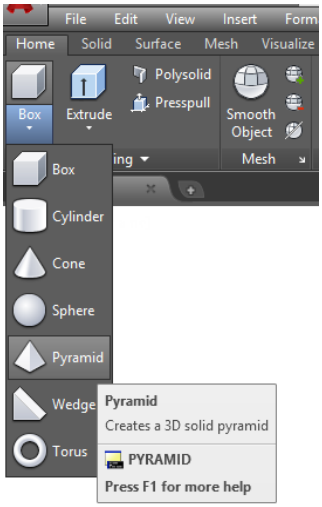
By default, a pyramid is defined by the center of the base point, a point on the middle of the edge, and another point that determines the height. Initially, the default base radius is not set to any value. During a drawing session, the default value for the base radius is always the previously entered base radius value for any solid primitive.

Use the Top Radius option to create a pyramid frustum.

Pyramid command of 3D Solid can be find out through following methods:

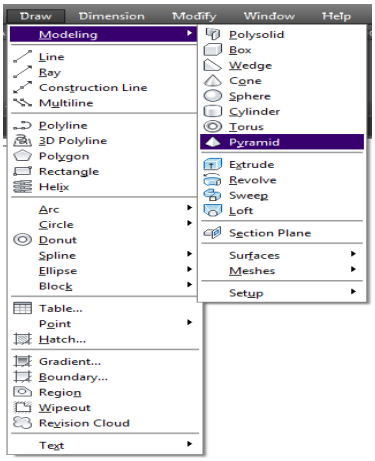
Through Ribbon

This figure describes the location of Pyramid command of 3D Solid in Home Tab.



Through Main Menu

Main Menu →Draw →Modeling →Pyramid



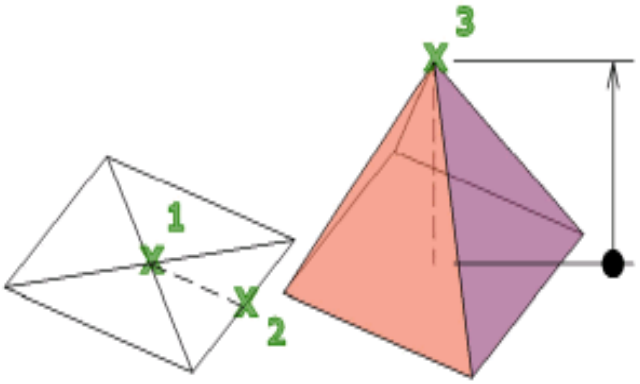
Do you know!

You can create a pyramid up to 32 sides.

Through Short Cut Key → Pyr enter

Application

Type **Pyr** at the Command Prompt → Specify center point of base→put the base radius value→Put the Value of Height→enter



Pyramid Option

The Pyramid option provides several methods for determining the size and rotation of the pyramids you create.

i. Sides

ii. Edges

iii. Frustum

iv. Axis Endpoint

v. Inscribed or circumscribed perimeter

i. Sides.

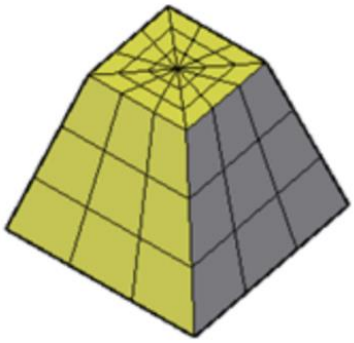
Use the Sides option to set the number of sides for the pyramid.

ii. Edges.

Use the Edges option to specify the dimension of the sides at the base.

iii. Frustum

Use the Top Radius option to create a frustum, which tapers to a planar face. The frustum face is parallel to and has the same number of sides as, the base.

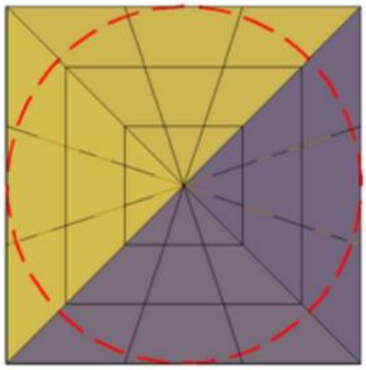


iv. Axis Endpoint

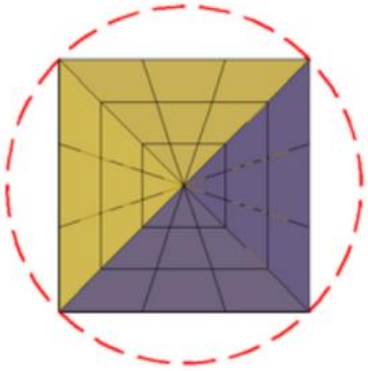
Use the Axis Endpoint option to specify the height and rotation of the pyramid. This endpoint is the top of the pyramid. The axis endpoint can be located anywhere in 3D space.

v. Inscribed or circumscribed perimeter.

Specify whether the pyramid base is drawn inside or outside of the radius.



Circumscribed



Inscribed

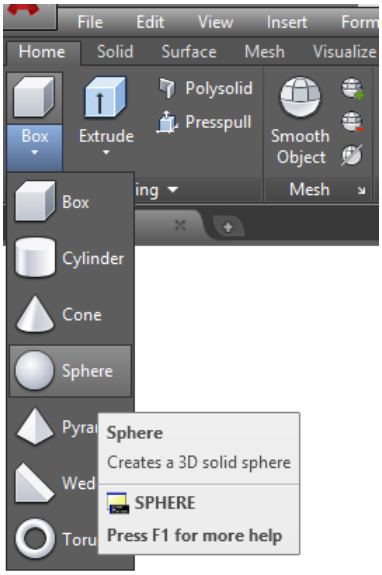
Sphere

You can create a torus by specifying the center, then the radius or diameter of the torus, and then the radius or diameter of the tube that surrounds the torus. You can control the smoothness of curved 3D solids, such as a torus, in a shaded or hidden visual style with the FACETRES system variable.

Sphere command of 3D Solid can be find out through following methods

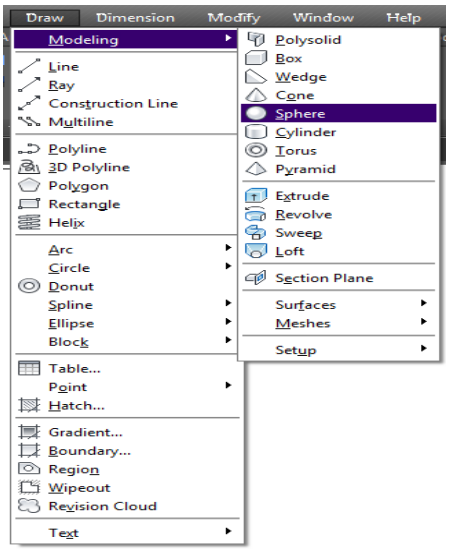
Through Ribbon

This figure describes the location of Sphere command of 3D Solid in Home Tab.



Through Main Menu

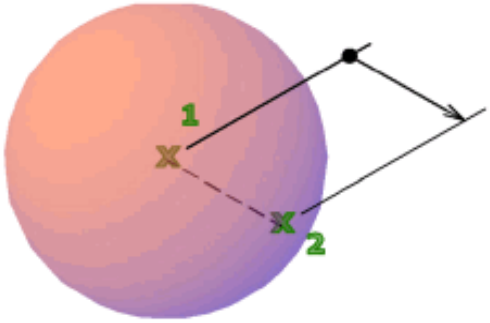
Main Menu →Draw →Modeling →Sphere



Through Short Cut Key → Sphere enter

Application

Type **Sphere** at the Command Prompt →Specify center point →Put the radius value→enter



Sphere Option

Use the following options to draw a sphere with the SPHERE command:

i. Center point

ii. 2p (Two Point)

iii. 3p (Three Point)

iv. TTR(Tangent, Tangent, Radius)

i. Center Point

Specify the center point of the sphere. When you specify the center point, the sphere is positioned so that its central axis is parallel to the Z axis of the current user coordinate system (UCS). Latitudinal lines are parallel to the XY plane.

iii. Radius Defines the radius of the sphere.

iv. Diameter Defines the diameter of the sphere.

ii. 3P (Three Points)

Define the circumference of the sphere by specifying three points anywhere in 3D space. The three specified points also define the plane of the circumference.

iii. 2P (Two Points)

Define the circumference of the sphere by specifying two points anywhere in 3D space. The plane of the circumference is defined by the Z value of the first point.

iv. TTR (Tangent, Tangent, Radius)

Define the sphere with a specified radius tangent to two objects. The specified tangency points are projected onto the current UCS.

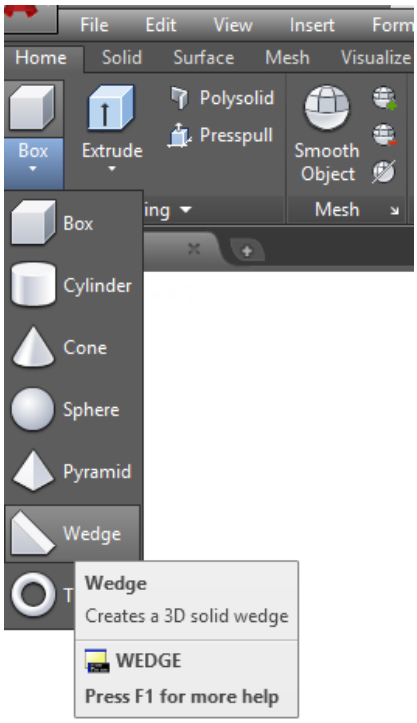
Wedge

The direction of the taper is always in the positive X-axis direction of the UCS. Entering a positive value draws the height along the positive Z axis of the current UCS. Entering a negative value draws the height along the negative Z axis

Wedge command of 3D Solid can be find out through following methods:

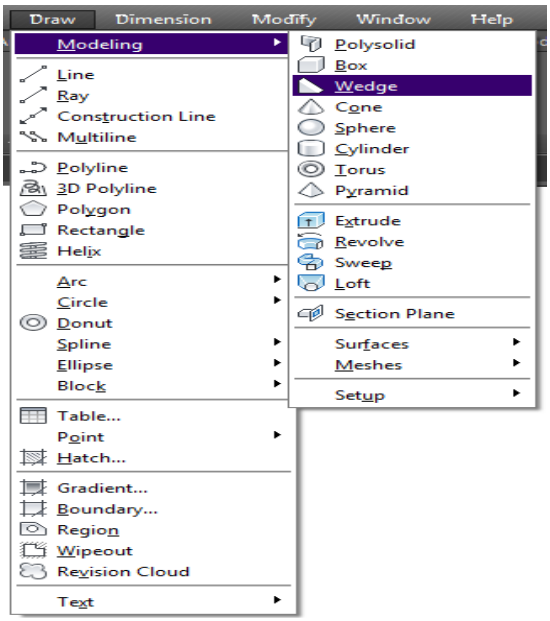
Through Ribbon

This figure describes the location of Wedge command of 3D Solid in Home Tab.



Through Main Menu

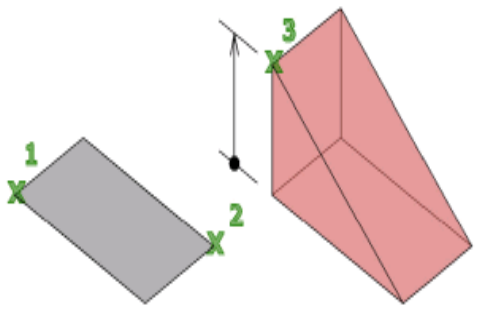
Main Menu →Draw →Modeling →Wedge



Through Short Cut Key → Wedge enter

Application

Type Wedge at the Command Prompt →Specify Start point→Put the Value of @ X and Y Co-ordinate→Put the Value of Height→enter



Wedge Option

The Wedge option provides several methods for determining the size and rotation of the wedges you create.

i. Center

ii. Cube

iii. Length

iv. 2 Point (With Height)

i. Center

Create the wedge by using a specified center point.

ii. Cube

Set all edges of the mesh wedge base to be of equal length.

iii. Length

Create a wedge with length, width, and height values you specify. The length corresponds to the X axis, the width to the Y axis, and the height to the Z axis. If you pick a point to specify the length, you also specify the rotation in the XY plane.

iv. 2 point (with Height)

Specify that the height of the wedge is the distance between the two specified points.

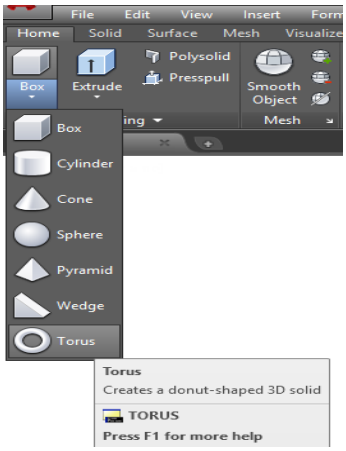
Torus

You can create a torus by specifying the center, then the radius or diameter of the torus, and then the radius or diameter of the tube that surrounds the torus. You can control the smoothness of curved 3D solids, such as a torus, in a shaded or hidden visual style with the FACETRES system variable

Torus command of 3D Solid can be find out through following methods:

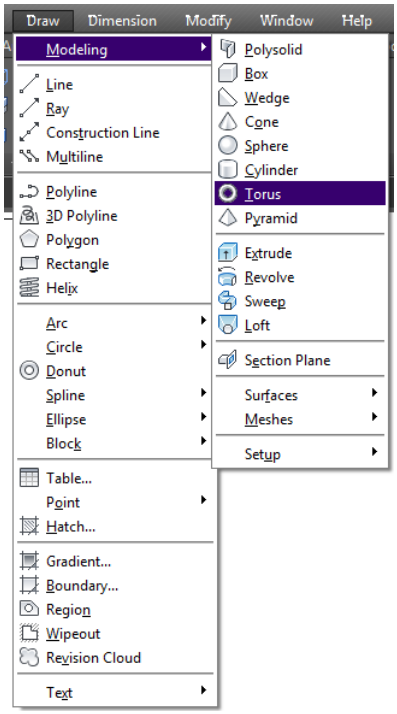
Through Ribbon

This figure describes the location of Torus command of 3D Solid in Home Tab.



Through Main Menu

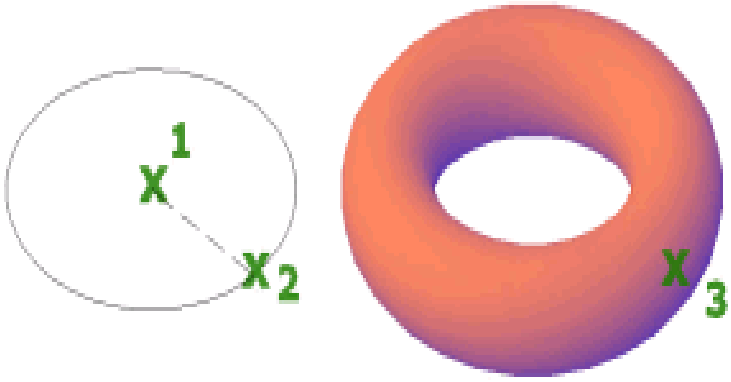
Main Menu →Draw →Modeling →Torus



Through Short Cut Key → Tor enter

Application

Type **Tor** at the Command Prompt → Specify center point of Torus→Put the radius value of Torus →Put the Tube radius Value →enter



Torus Option

The Torus option provides several methods for Determining the size and rotation of the mesh tours you create.

i. 3P

ii. 2p

iii. TTR

iv. Radius

i. 3P (Three Points)

Use the 3P (Three Points) option to define the size of the torus anywhere in 3D space. The three points also define the plane of the circumference. Use this option to rotate the torus as you create it.

ii. 2P (Two Points)

Use the 2P (Two Points) option to define the size of the torus anywhere in 3D space. The plane of the circumference matches the Z value of the first point.

iii. TTR (Tangent, Tangent, Radius)

Use the TTR (Tangent, Tangent, and Radius) option to define points on two objects. Depending on the specified radius distance, the path of the torus is located as near as possible to the tangent points you specify. You can set up tangency with circles, arcs, lines, and some 3D objects. The tangency points are projected into the current UCS.

iv. Radius

Define the radius of the torus: the distance from the center of the torus to the center of the tube. A negative radius creates a solid shaped like an American football.

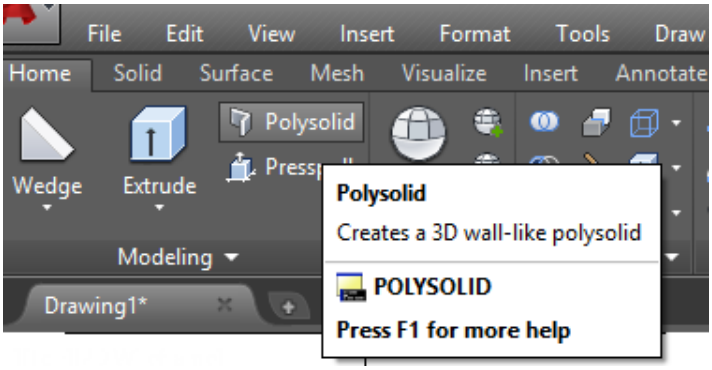
2. Poly Solid Command

You can create walls with straight and curved segments of constant height and width. With the POLYSOLID command, you can convert an existing line, 2D polyline, arc, or circle to a solid with a rectangular profile. A polysolid can have curved segments, but the profile is always rectangular by default. You can draw a solid with POLYSOLID just as you would a polyline. The PSOLWIDTH system variable sets the default width for the solid. The PSOLHEIGHT system variable sets the default height for the solid.

Poly Solid command of 3D Solid can be find out through following methods:

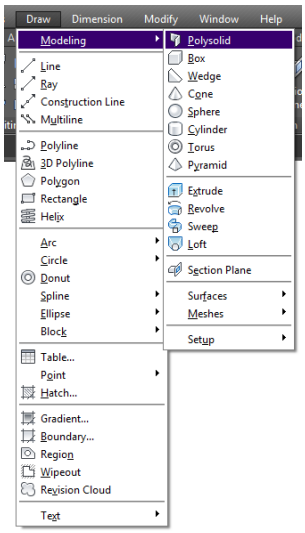
Through Ribbon

This figure describes the location of poly Solid command of 3D Solid in Home Tab.



Through Main Menu

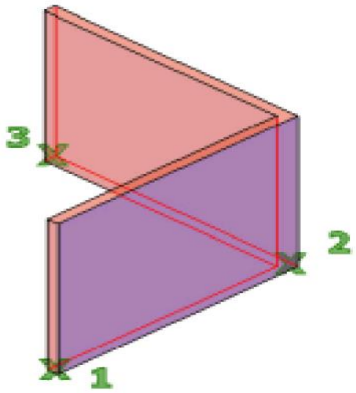
Main Menu →Draw →Modeling →PolySoild



Through Short Cut Key → Ploysoild enter

Application

Type **polysolid** at the Command Prompt → Specify start point or [Object/Height/Width/Justify] <Object>→ Specify next point or [Arc/Undo]→ Specify next point or [Arc/Undo] →enter



POLY SOLID OPTIONS

Use the following options to control the size and shape of the polysolid you create

i. Object
ii. Height
iii. Width
iv. Justify

i. Object
Specify an object to convert to a solid. You can convert:

- Line
- Arc
- 2D polyline
- Circle

ii. **Height**
Specify the height of the solid

iii. **Width**
Specify the width of the solid.

iv. **Justify**
Set the width and height as the solid to be left, right, or center justified when defining the profile with the command. The justification is based on the starting direction of the first segment of the profile.

3. Extrude Command

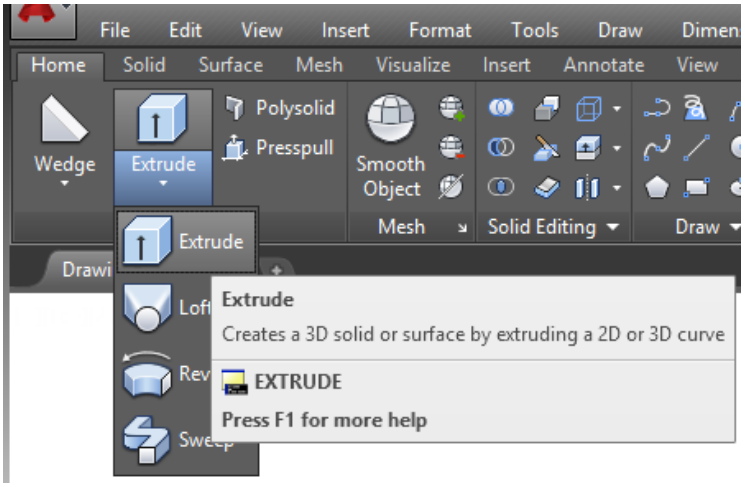
Extrusions can extend in the Z direction or be set to taper or follow a path.

You can extrude an open or closed object to create 3D surface or solid

EXTRUDE command of 3D Solid can be found out through following methods:

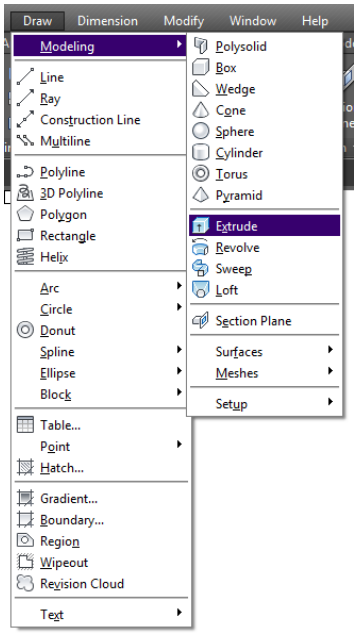
Through Ribbon

This figure describes the location of EXTRUDE command of 3D Solid in Home Tab.



Through Main Menu

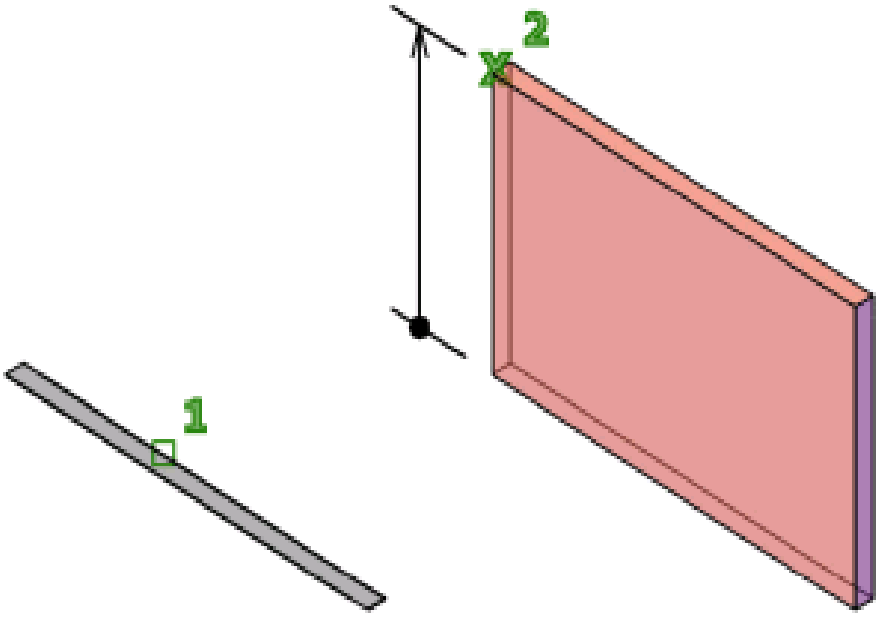
Main Menu →Draw →Modeling → EXTRUDE



Through Short Cut Key → Ext enter

Application

Type **Ext** at the Command Prompt → Select Object→enter→ Specify height of extrusion or [Direction/Path/Taper angle] →enter



First of all draw all the 2D objects in ploy line.

There are two methods to convert a line into a poly line.

1- **Boundary Command**

   This command use for 2D close object converts in to poly line

2- **Polyline Edit Command**

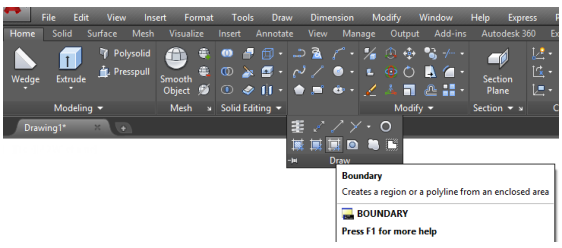
   This command for 2D open object converts in to poly line

1. Boundary Command

Create a region or a polyline from an enclosed area.
Boundary command can be found out through following methods:

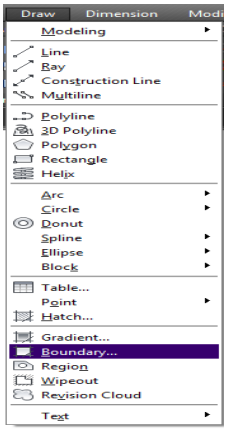
Through Ribbon

This figure describes the location of Boundary command in Home Tab.



Through Main Menu

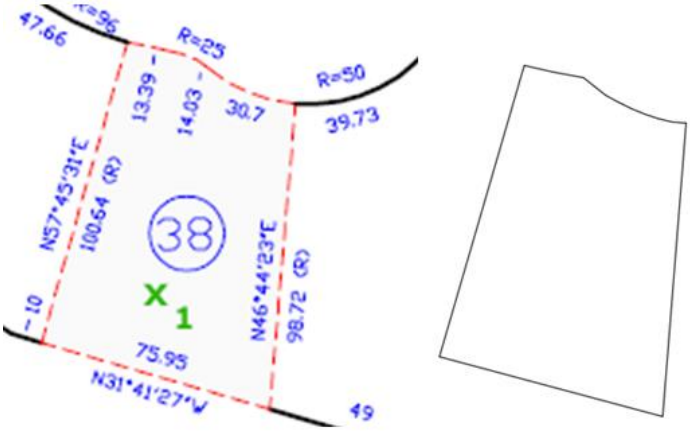
Main Menu →Draw →Boundary



Through Short Cut Key → Bo enter

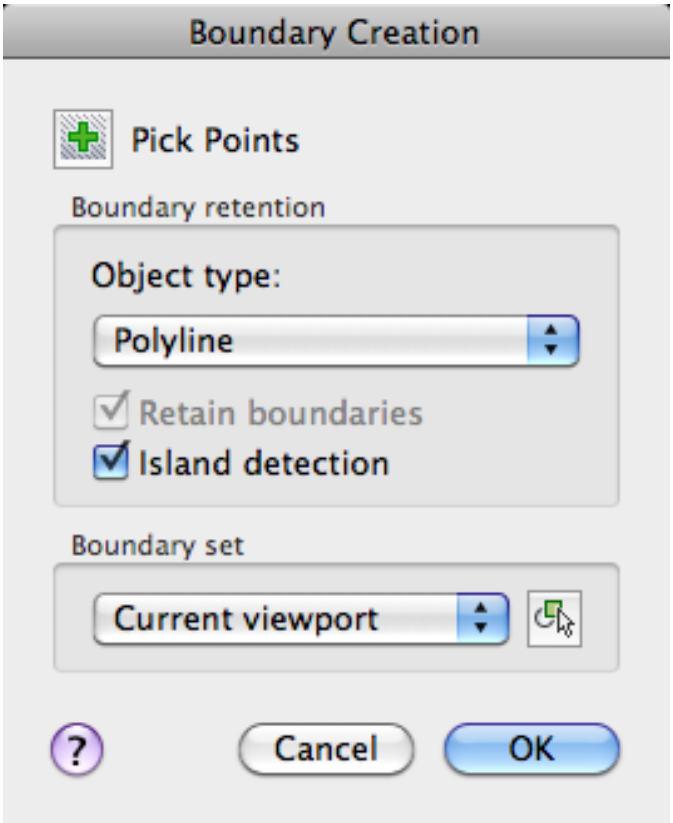
Application

Type Bo at the Command Prompt → enter→ Open Boundary Window →pick point →enter



Boundary Creation Dialog Box

Define the object type, boundary set, and island detection method for creating region or polyline using a specified point within an area enclosed by objects.



List of Options

The following options are displayed.

Pick Points

Determine a boundary from existing objects that form an enclosed area around the specified point.

Boundary Retention

Define the type of boundary object that will be created from the enclosed area and how enclosed areas are detected.

- **Object Type**
  Control the type of the new boundary object. Boundary creates the boundary as a region or a polyline object.
- **Retain Boundaries**
  Control whether the objects created are retained after the dialog box is closed.
- **Island Detection**
  Control whether boundary detects internal closed boundaries, called islands.

Boundary Set

Defines the set of objects boundary analyzes when defining a boundary from a specified point.

Current Viewport

Defines the boundary set from everything in the current viewport extents. Selecting this option discards any current boundary set.

2. Polyline Edit Command

Common uses for PEDIT include joining 2D polylines, converting lines and arcs into 2D polylines, and converting polylines into curves that approximate B-splines (spline-fit polylines).

- **Short Cut Key** → Pe enter
- **Application**

Type **Pe** at the Command Prompt → enter→ Select polyline → enter→ J enter (For Joint the line) → Select all lines →enter

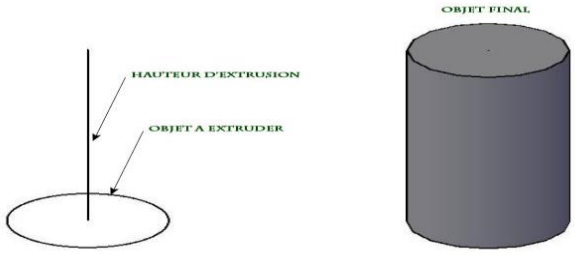
Extrude Command Options

a) Height of extrusion
b) Direction
c) Path
d) Taper angle

Note:
Select face and edge sub objects by pressing Ctrl while you select them.

a) Height of extrusion

Extrudes objects along the positive *Z* axis of the object's coordinate system if you enter a positive value. If you enter a negative value, the objects are extruded along the negative *Z* axis. Objects do not have to be parallel to the same plane. If all the objects are on a common plane, the objects are extruded in the direction of the normal of the plane.

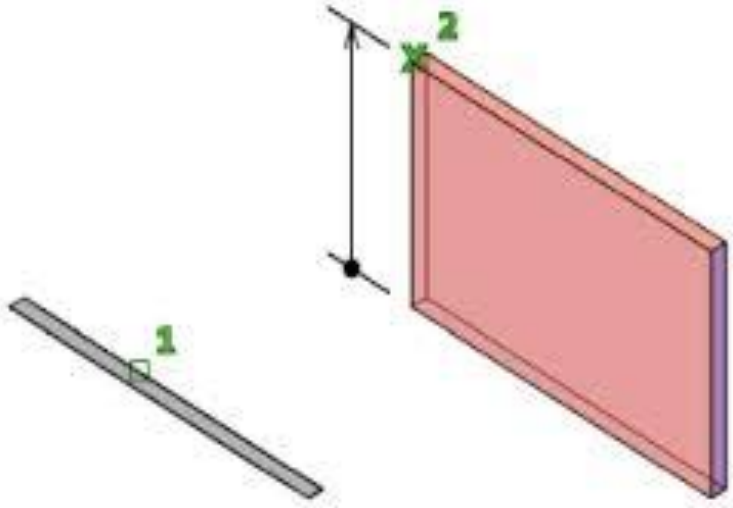


b) Direction

Specifies the length and direction of the extrusion with two points you specify. (The direction cannot be parallel to the plane of the sweep curve created by the extrusion.)

Do you know!

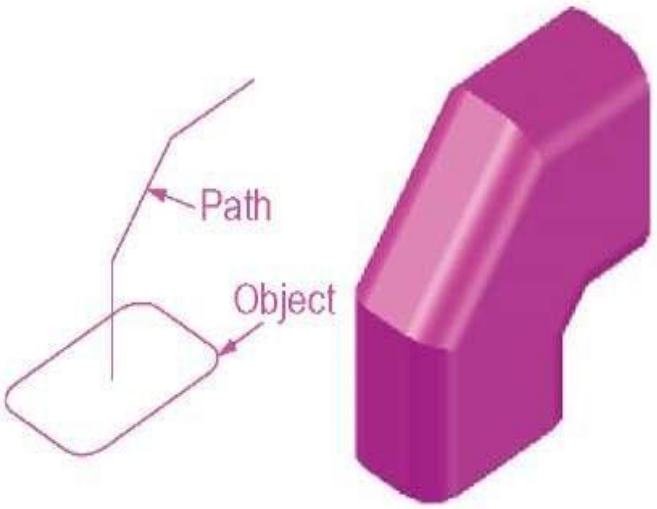
By default, planar objects are extruded in the direction of the object's normal



- Start point of direction. Specifies the first point in the direction vector.
- End point of direction. Specifies the second point in the direction vector.

c) Path

Specifies the extrusion path based on an object that you select. The path is moved to the centric of the profile. Then the profile of the selected object is extruded along the chosen path to create solids or surfaces.



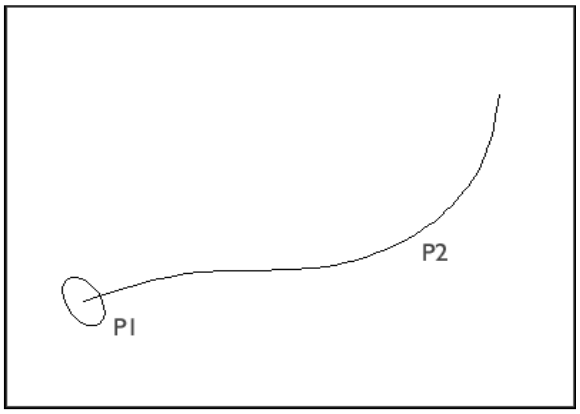
The path should not lie on the same plane as the object, nor should the path have areas of high curvature.

The extrusion starts from the plane of the object and maintains its orientation relative to the path. If the path contains segments that are not tangent, the program extrudes the object along each segment and then miters the joint along the plane bisecting the angle formed by the segments. If the path is closed, the object should lie on the miter plane. This allows the start and end sections of the solid to match up. If the object is not on the miter plane, the object is rotated until it is on the miter plane. Objects with multiple loops are extruded so that all the loops appear on the same plane at the end section of the extruded solid.

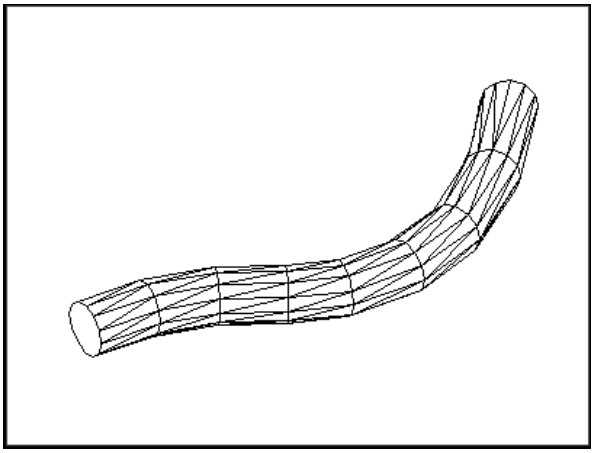
Application

Type **Ext** at the Command Prompt → Select Object→enter→ Specify height of extrusion or [Direction/Path/Taper angle] →**P enter (**For Path) →Select Path→ enter

Circle Extruded Along a Path



Hidden Line Removal of Extruded Circle



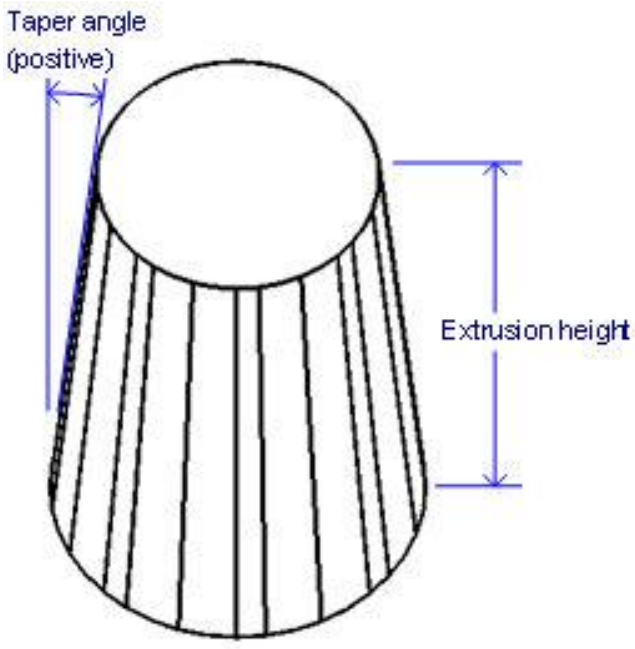
d) Taper Angle

Positive angles taper in from the base object. Negative angles taper out. The default angle, 0, extrudes a 2D object perpendicular to its 2D plane. All selected objects and loops are tapered to the same value. Specifying a large taper angle or a long extrusion height can cause the object or portions of the object to taper to a point before reaching the extrusion height. Individual loops of a region are always extruded to the same height. When an arc is part of a tapered extrusion, the angle of the arc remains constant, and the radius of the arc changes.

- Angle of taper. Sets an angle between -90 and +90 degrees.
- Specify two points. Sets the taper angle based on two points you specify in the drawing area. The taper angle applied to the extrusion is the distance between the two specified points.

Application

Type **Ext** at the Command Prompt → Select Object→enter→ Specify height of extrusion or [Direction/Path/Taper angle] →**T enter(**For Taper Angle) →Put the Value of Taper Angle→Specify height of extrusion or [Direction/Path/Taper angle] →put the Value of Height→ enter



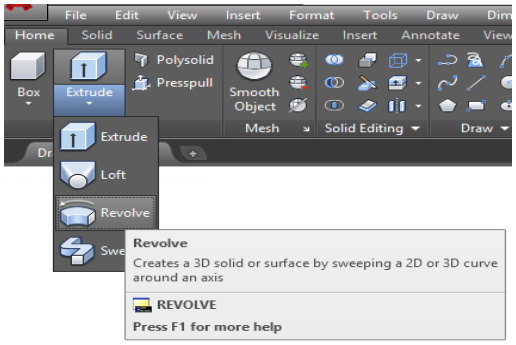
4. Revolve Command

Open profiles create surfaces and closed profiles can create either a solid or a surface. The Mode option controls is a solid of surface is created. When creating a surface. You cannot revolve objects contained within a block or objects that will self-intersect. REVOLVE ignores the width of a polyline and revolves from the center of the path of the polyline. The right-hand rule determines the positive direction of rotation. For more information, see Understand the UCS in 3D.

Revolve command of 3D Solid can be found out through following methods.

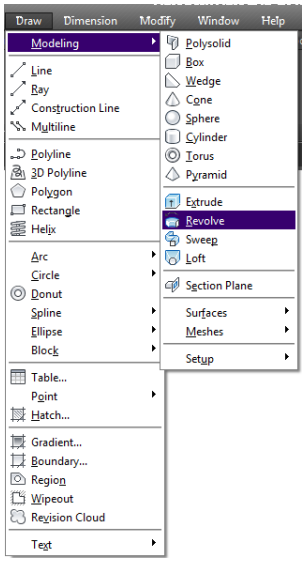
Through Ribbon

This figure describes the location of Revolve command of 3D Solid in Home Tab.



Through Main Menu

Main Menu →Draw →Modeling → Revolve

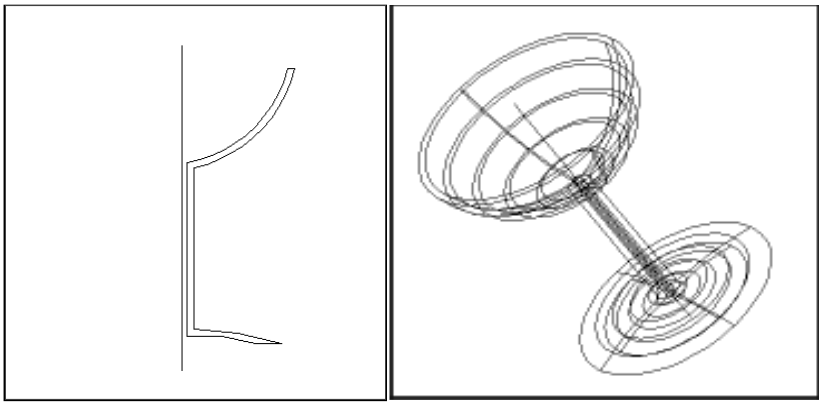


Through Short Cut Key → Rev enter

Application

Type **Rev** at the Command Prompt →Select Object→enter→ Specify start point for axis of revolution →

Select first point of axis →Select 2nd point of axis→ Specify angle of revolution <360>→put the revolving angle → enter



Remember

If you revolve a closed object, the result is a solid. If you revolve an open object, the result is a surface.

Revolve Options

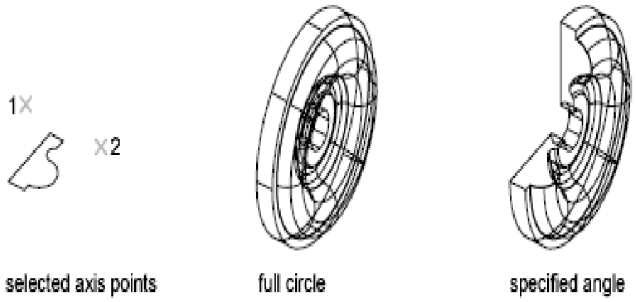
i. Mode
ii. Start Angle
iii. Reverse
iv. Expression

i. Mode

Control whether the revolve action creates a solid or a surface. Surfaces are extended as either NURBS surfaces or procedural surfaces, depending on the SURFACEMODELINGMODE system variable

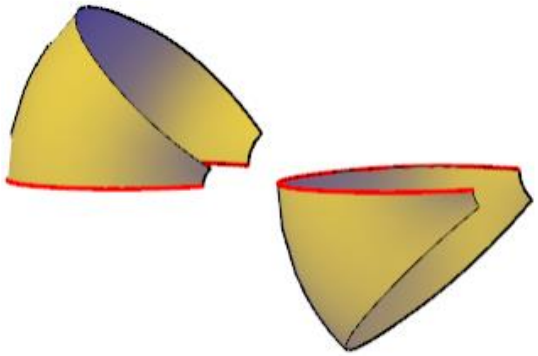
ii. Start Angle

Specify an offset for the revolution from the plane of the object being revolved. Drag your cursor to specify and preview the start angle of the object.



iii. Reverse

Change the direction of the Revolve; similar to entering a - (minus) angle value. The revolved object on the right shows a spline revolved at the same angle as the object on the left, but using the reverse option



iv. Expression

Enter a formula or equation to specify the revolve angle. See Constrain a Design with Formulas and Equations.

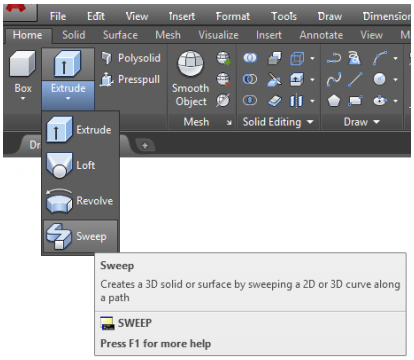
5. Sweep Command

Creates a solid or surface by sweeping an open or closed, planar or non-planar curve (profile) along an open or closed path. Open curves create surfaces and closed curves create solids or surfaces, depending on the specified mode. You can use the following objects and paths when creating a swept solid or surface:

Sweep command of 3D Solid can be found out through following methods.

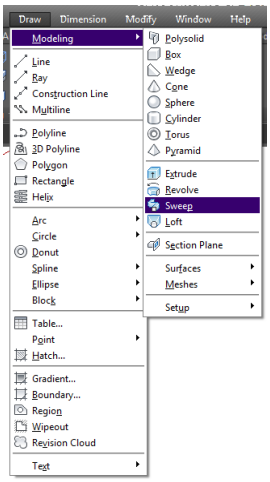
Through Ribbon

This figure describes the location of Sweep command of 3D Solid in Home Tab.



Through Main Menu

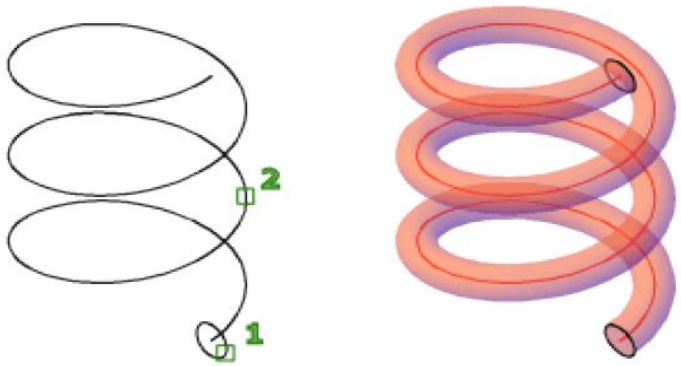
Main Menu →Draw →Modeling → Sweep



Through Short Cut Key → Sweep enter

Application

Type **Sweep** at the Command Prompt →Select Object to sweep→enter→ Select sweep path → enter



Sweep Options

i. Alignment

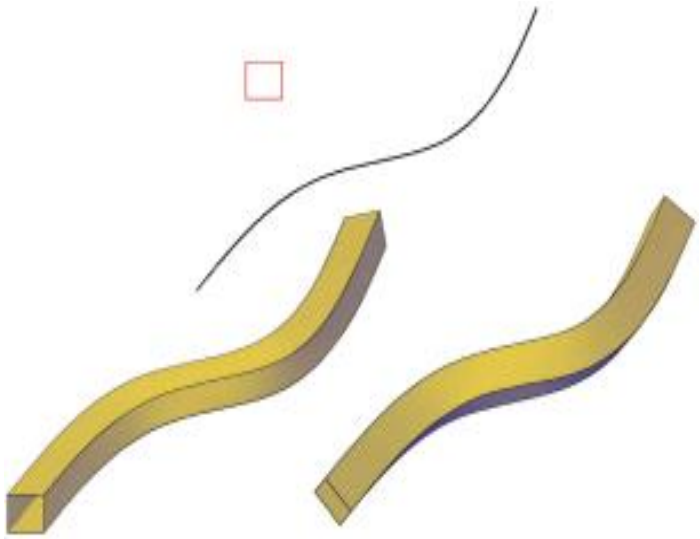
ii. Base point

iii. Scale

iv. Twist

i. Alignment

Specify whether the profile is aligned to be normal to the tangent direction of the sweep path. If the profile is not perpendicular (normal) to the tangent of the start point of the path, then the profile automatically aligns. Enter No at the alignment prompt to prevent this.



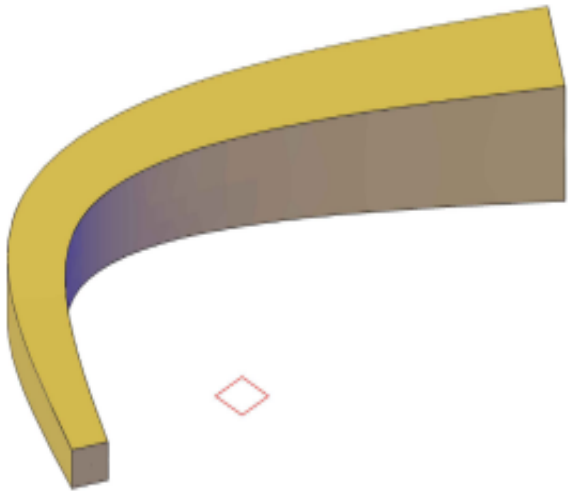
ii. Base point

Specify a base point for the objects to be swept.

iii. Scale

Specify a scale factor for a sweep operation. The scale factor is uniformly applied to the objects that are swept from the start to the end of the sweep path.

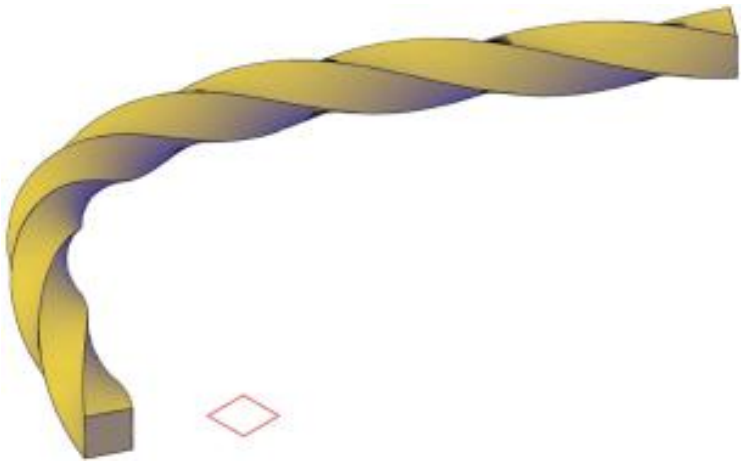
Reference. Scale the selected objects based on the length you reference by picking points or entering values.



iv. Twist

Set a twist angle for the objects being swept. The twist angle specifies the amount of rotation along the entire length of the sweep path.

Specify whether the curve(s) being swept will naturally bank (rotate) along a 3D sweep path (3D polyline, spline, or helix).



6. Loft Command

LOFT command creates a solid by joining several crass sections lying on different planes. The cross sections define the profile (Shape) of the resulting solid. Cross sections (generally, curves or circle).Loft draws a solid in

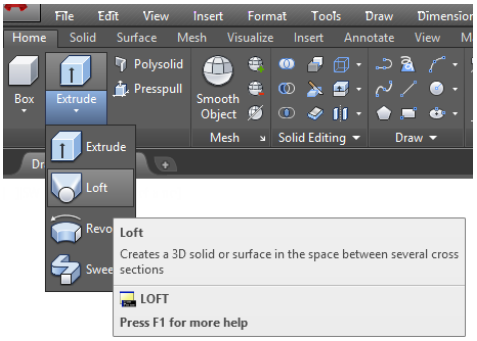
the space among the different cross sections. You must specify at least two cross sections when you use the LOFT command.

Create 3D solid by specifying a series of cross sections. The cross sections define the shape of the resulting solid. You must specify at least two cross sections. Loft profiles can be open or closed, planar or non-planar, and can also be edge sub objects. Use the mode option to select whether to create a solid.

Loft command of 3D Solid can be found out through following methods.

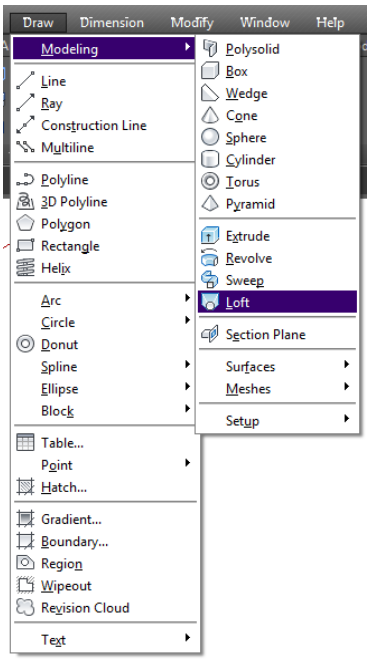
Through Ribbon

This figure describes the location of Loft command of 3D Solid in Home Tab.



Through Main Menu

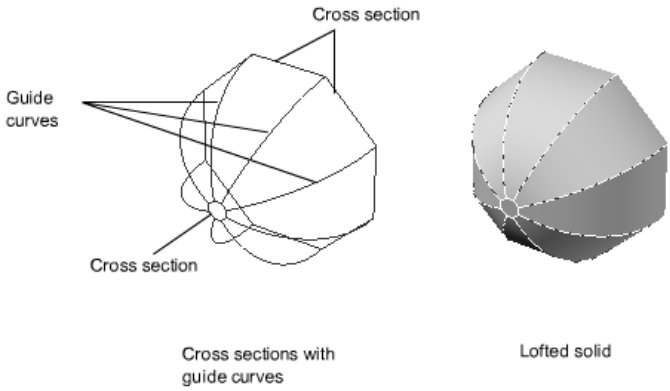
Main Menu →Draw →Modeling → Loft



Through Short Cut Key → Loft enter

Application

Type **Loft** at the Command Prompt → Select cross-sections in lofting order →Select Cross sections→ enter

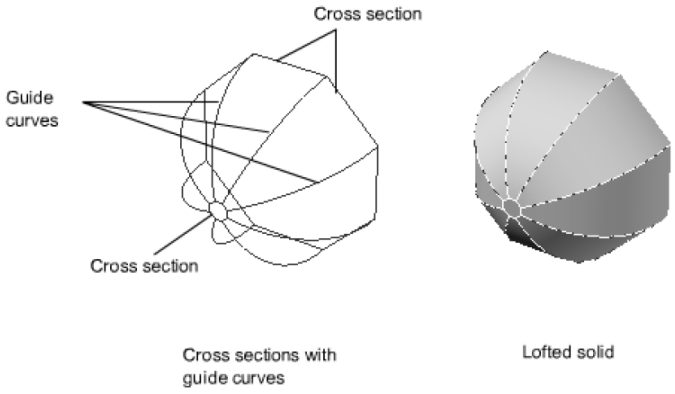


Loft Command Options

i. Guides
ii. Path
iii. Cross sections only
iv. Settings

i. Guides

Specifies guide curves that control the shape of the lofted solid or surface. You can use guide curves to control how points are matched up on corresponding cross sections to prevent undesired results, such as wrinkles in the resulting solid or surface.

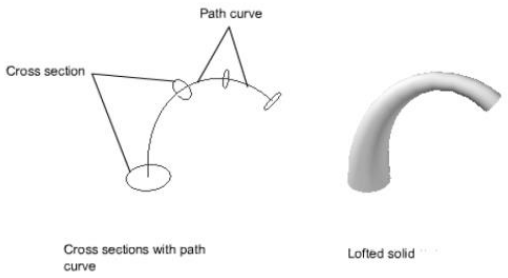


- Intersects each cross section
- Starts on the first cross section
- Ends on the last cross section

Select any number of guide curves for the lofted surface or solid and press Enter.

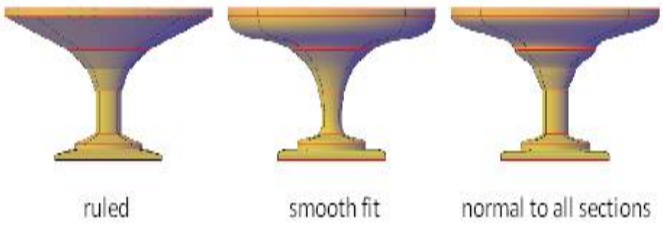
ii. Path

Specify a path for the loft operation to obtain more control over the shape of the lofted object. For best results, start the path curve on the plane of the first cross section and end it on the plane of the last cross section.



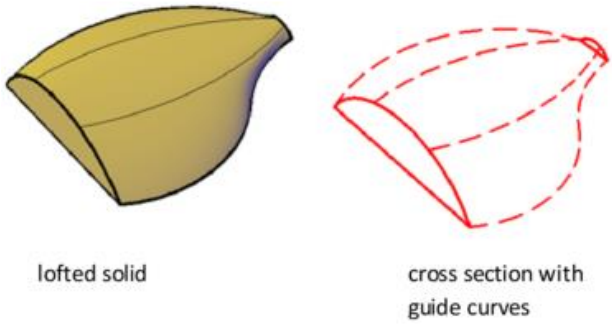
iii. Cross-section

Select a series of cross-section profiles to define the shape of the new 3D object. Lofted objects with different cross-section settings as you create a lofted object; you can adjust its shape by specifying how the profile passes through the cross sections (for example, a sharp or smooth curve). You can also modify the settings later in the Properties Inspector.



Guide Curves.

Specify guide curves to match points on corresponding cross sections. This method prevents undesired results, such as wrinkles in the resulting 3D object.



Each guide curve must meet the following criteria

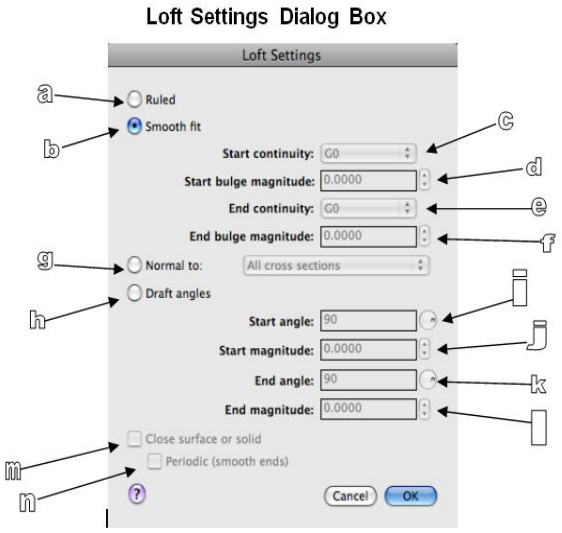
- Intersects each cross section
- Starts on the first cross section
- Ends on the last cross section

iv. Settings

Displays the **Loft Settings dialog box**.

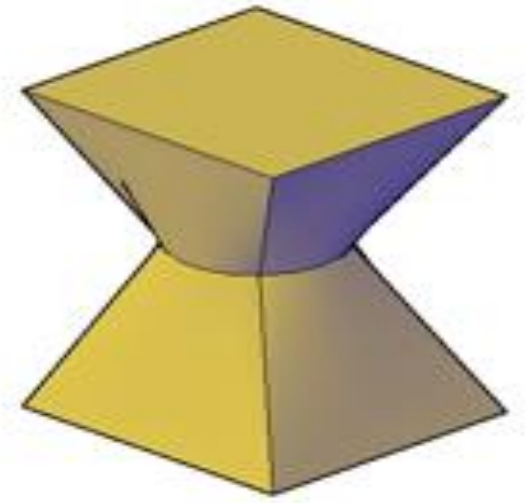
Control the contour of a lofted surface at its cross sections.

Also allows you to close the surface or solid.



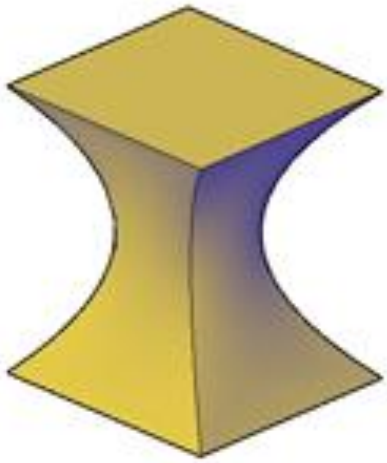
a) Ruled

Specifies that the solid or surface is ruled (straight) between the cross sections and has sharp edges at the cross sections



b) Smooth Fit

Specifies that a smooth solid or surface is drawn between the cross sections and has sharp edges at the start and end cross sections.



c) Start Continuity

Set the tangency and curvature of the first cross section.

d) Start Bulge Magnitude

Set the size of the curve of the first cross section.

e) End Continuity

Set the tangency and curvature of the last cross section.

f) End Bulge Magnitude

Set the size of the curve of the last cross section.

g) Normal To

Controls the surface normal of the solid or surface where it passes through the cross sections

a. **Start Cross Section**
   Specify that the surface normal is normal to the start cross section.
b. **End Cross Section**
   Specify that the surface normal is normal to the end cross section.
c. **Start and End Cross Sections**
   Specify that the surface normal is normal to both the start and end cross sections.

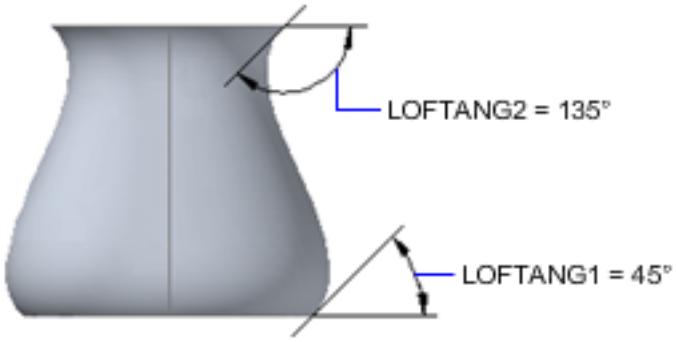
d. **All Cross Sections**

Specify that the surface normal is normal to all cross sections.

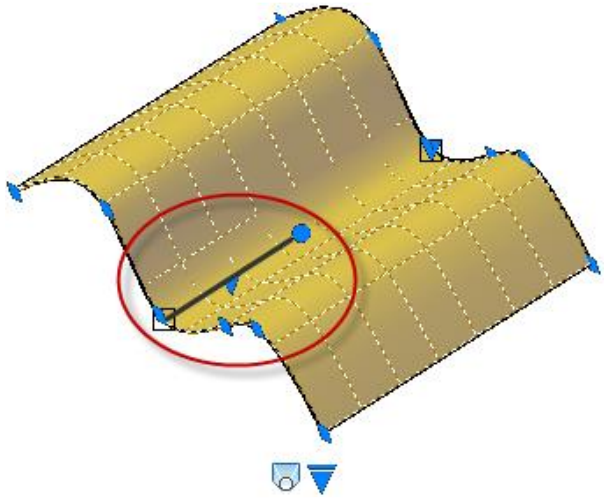
h) Draft Angles

Control the draft angle and magnitude of the first and last cross sections of the lofted solid or surface. The draft angle is the beginning direction of the surface. 0 is defined as outward from the plane of the curve.

The following illustration shows the affect of using a different draft angle for the first and last cross sections of a lofted solid. The first cross section is assigned a draft angle of 45 degrees, while the last cross section is assigned a draft angle of 135 degrees.



You can also use the draft angle handle to adjust the draft angle (triangular grip) and magnitude (circular grip).



i) Start Angle

Specify the draft angle for the start cross section.

j) Start Magnitude

Control the relative distance of the surface from the start cross section in the direction of the draft angle before the surface starts to bend toward the next cross section.

k) End Angle

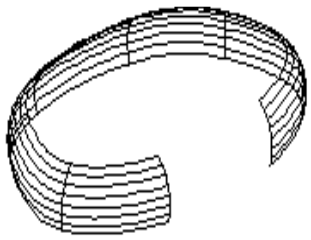
Specify the draft angle for the end cross section.

l) End Magnitude

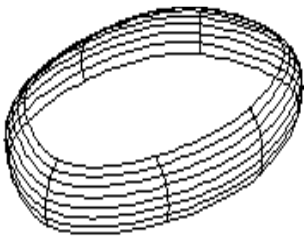
Control the relative distance of the surface from the end cross section in the direction of the draft angle before the surface starts to bend toward the previous cross section,

m) Close Solid

Close and opens a surface or solid. When using this option, the cross sections should form a torus-shaped pattern so that the lofted solid can form a closed tube.



loft created with Close surface solid option unchecked



loft created with Close surface solid option checked

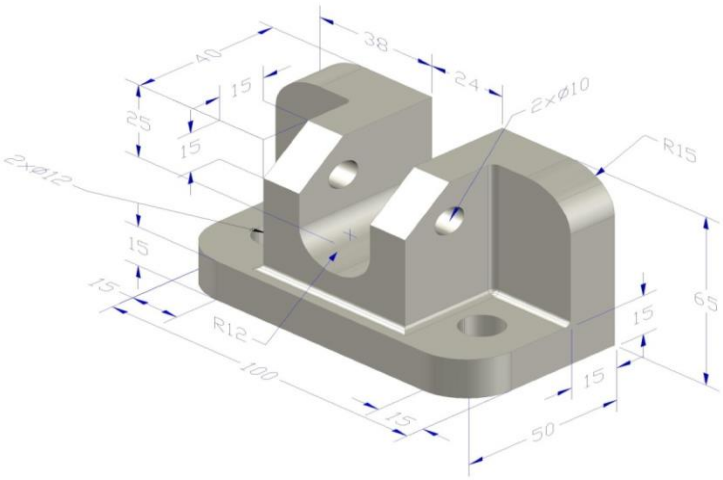
n) Periodic (Smooth Ends)

Create a smooth, closed surface whose seam will not kink if it is reshaped. This option is only available if the loft is ruled or smooth fit and the Close Surface or solid option is selected.

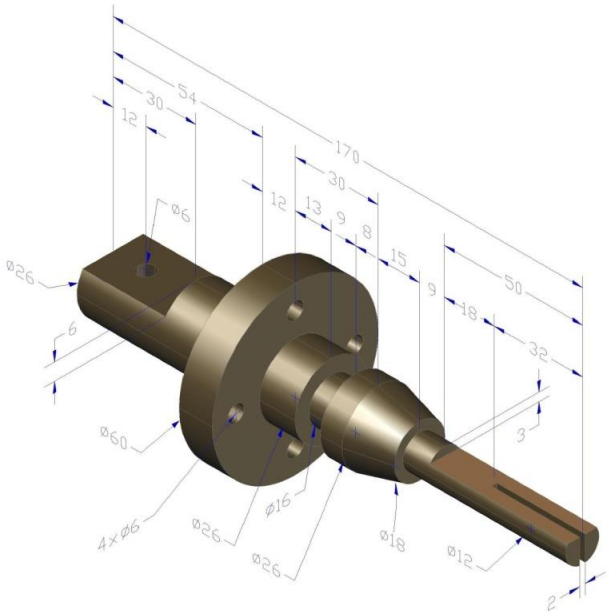
Learner’s Activity

Use to Solids command to draw the flowing object

Assignment # 01



Assignment # 02



Learning Unit 2: Edit Effectively 3D Solid Object

Overview

In this learning unit, you will learn how to edit 3D solid object. After completion of this learning unit, you will be able to edit solids. You can then combine these shapes to create more complex solids.

In this unit, you will learn different commands used for editing of a 3D solid object. At the end of this learning unit, you will be able to edit 3D Solids.

1. Convert to Solid

Converts 3D meshes and polylines and circles with thickness to 3D solids. Take advantage of the solid modeling capabilities available for 3D solids. When you convert mesh, you can specify whether the converted objects are smoothed or faceted, and whether the faces are merged.

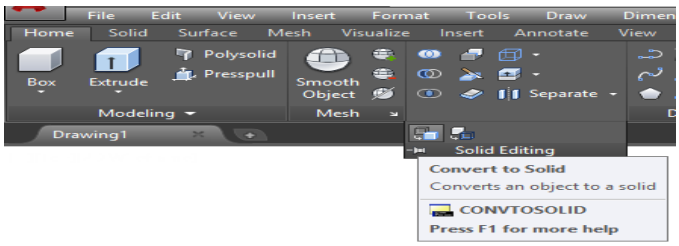
The following objects that can be converted to solid:

- **Mesh**
- **Polyline**
- **Surface**

Convert to Solid command of 3D Solid can be found out through following methods.

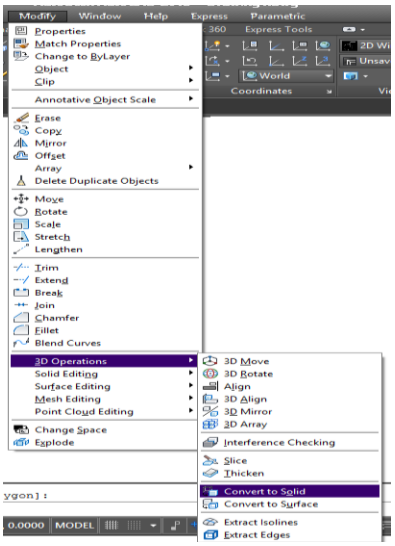
Through Ribbon

This figure describes the location of Convert to Solid command of 3D Solid in Home Tab.



Through Main Menu

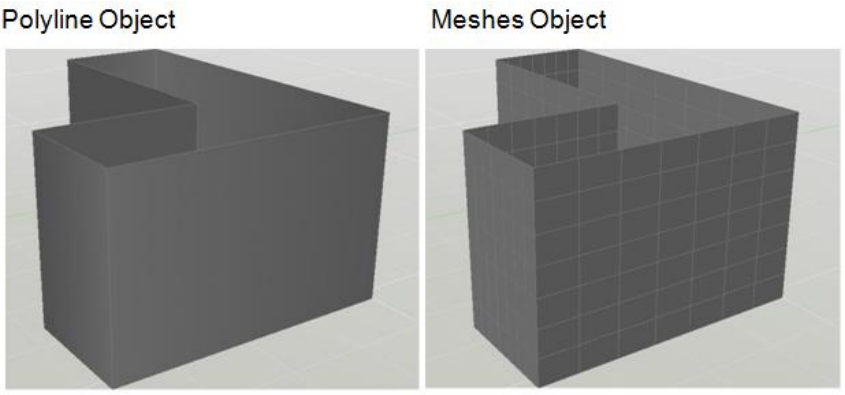
Main Menu →Modify→3D Operations → Convert to Solid

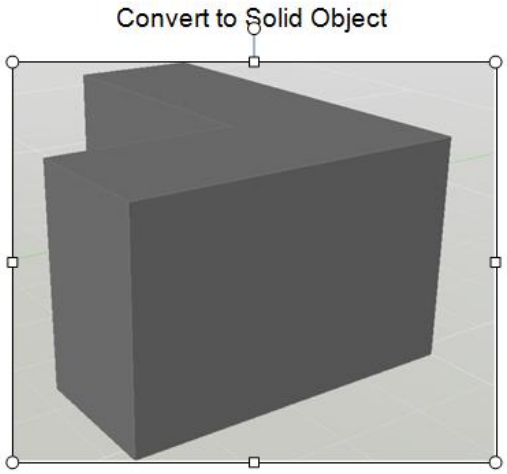


Through Short Cut Key → CONVTOSOLID enter

Application

Type **CONVTOSOLID** at the Command Prompt → Select Object to Solid→ enter





2. Convert to Surfaces

As you convert objects to surfaces, you can specify whether the resulting object is smooth or faceted

With the CONVTOSURFACE command, you can convert the following objects into surfaces

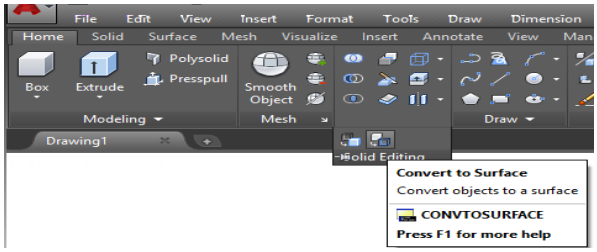
Note:
You can create surfaces from 3D solids with curved faces, such as a cylinder, with the EXPLODE command

- **2D solids**
- **3D solids**
- **Regions**
- **Open, zero-width polylines with thickness**
- **Lines with thickness**
- **Arcs with thickness**
- **Mesh objects**
- **Planar 3D faces**

Convert to Surface command of 3D Solid can be found out through following methods.

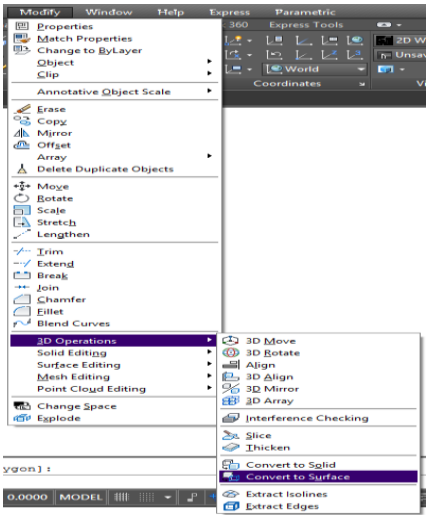
Through Ribbon

This figure describes the location of Convert to Surface command of 3D Solid in Home Tab.



Through Main Menu

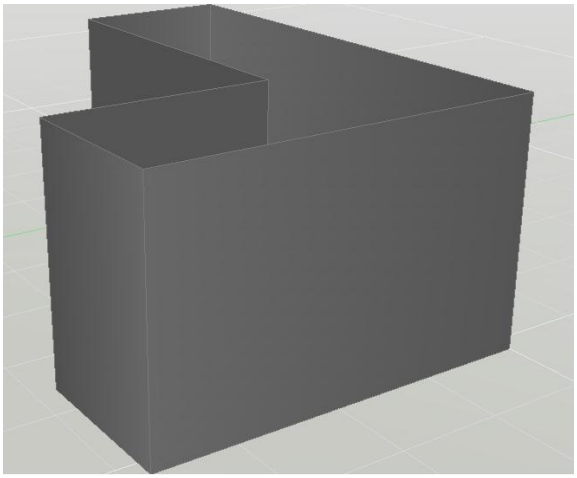
Main Menu→Modify→3D Operations→ Convert to Surface

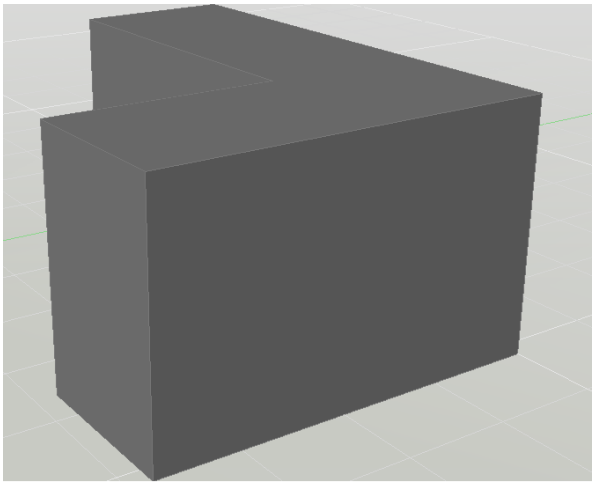


Through Short Cut Key → CONVTOSURFACE enter

Application

Type **CONVTOSURFACE** at the Command Prompt → Select object s → enter





Note:
Objects that can be converted to 3D Surfaces
Objects
2D solids, 3D solids
Regions, Open, zero-width polylines with thickness
Lines with thickness
Arcs with thickness
Mesh objects, planar 3D faces

3. Solid Editing Commands

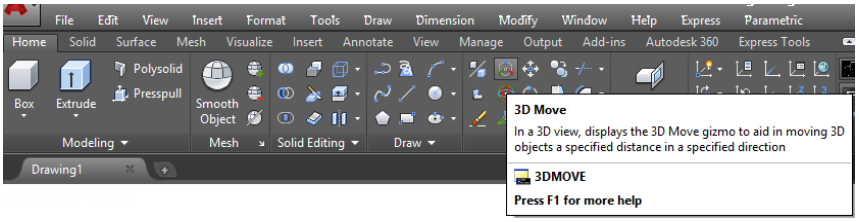
1. 3D Move Command

With the 3D Move gizmo, you can move selected objects and sub objects freely or constrain the movement to an axis or plane. The 3D Move gizmo is displayed whenever you select an object in a view with a 3D visual style. If you are working in a viewport with 2D Wireframe set as the visual style, 3DMOVE temporarily changes the visual style to 3D Wireframe for the duration of the command.

3D Move command for 3D Solid Objects can be found out through following methods.

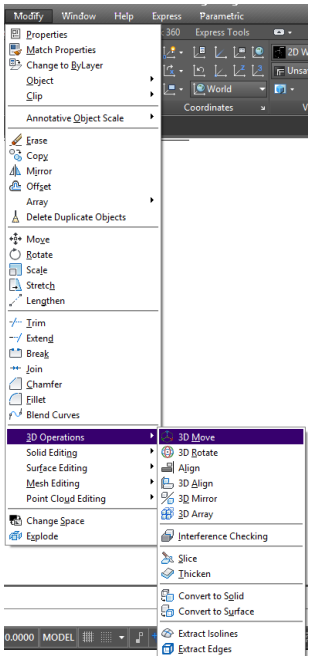
Through Ribbon

This figure describes the location of 3D Move command for 3D Solid objects in Home Tab.



Through Main Menu

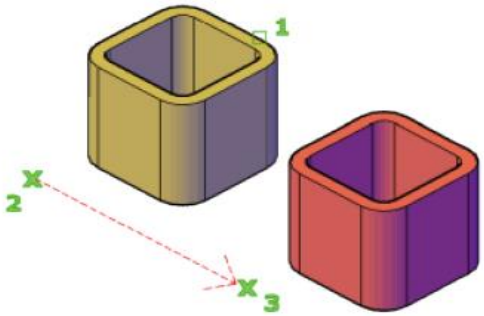
Main Menu→Modify→3D Operations→ 3D Move



Through Short Cut Key → 3Dmove enter

Application

Type **3Dmove** at the Command Prompt → Select object → enter → Specify base point or [Displacement] →Specify displacement Value →enter



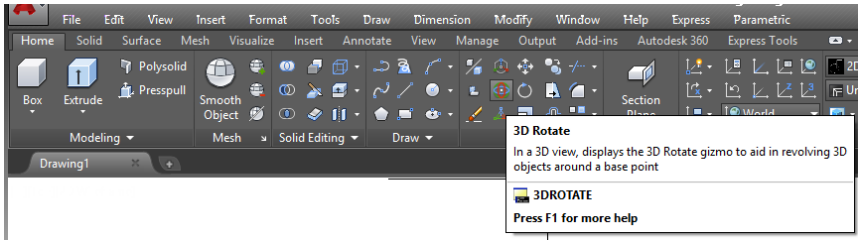
2. 3D Rotate Command

With the 3D rotate gizmo, you can rotate selected objects and sub objects freely or constrain the rotation to an axis. If you are working in a viewport with 2D Wireframe set as the visual style, 3DROTATE temporarily changes the visual style to 3D Wireframe for the duration of the command.

3D Rotate command for 3D Solid Objects can be found out through following methods.

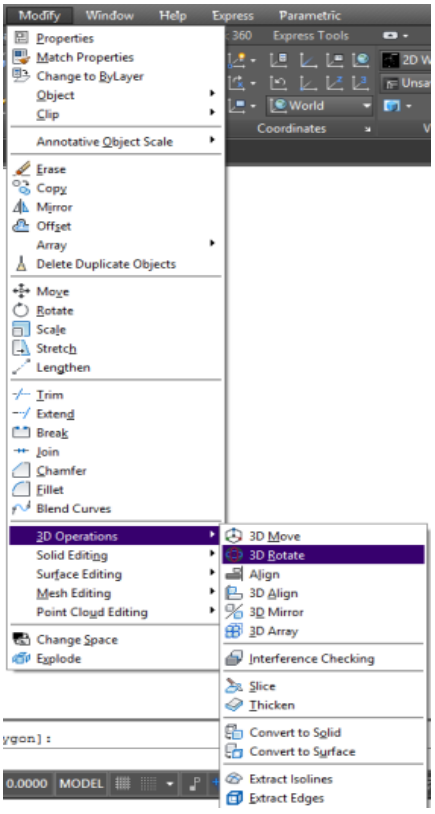
Through Ribbon

This figure describes the location of 3D rotate command for 3D Solid objects in Home Tab.



Through Main Menu

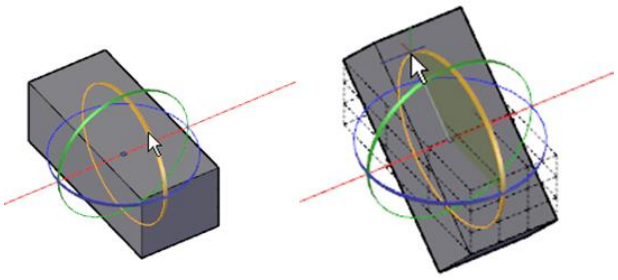
Main Menu→Modify→3D Operations→ 3D Rotate



Through Short Cut Key → 3Drotate enter

Application

Type **3Drotate** at the Command Prompt → Select object →enter→Specify base point→Pick a rotation axis→ Specify angle rotation value → enter



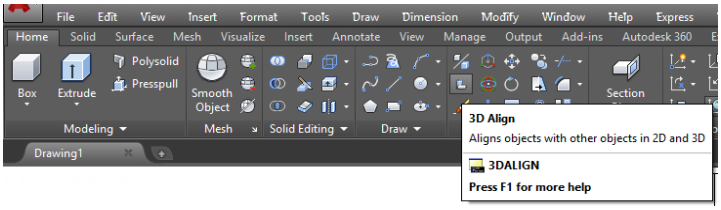
3. 3D Align Command

You can specify one, two, or three points for the source object. Then, you can specify one, two, or three points for the destination.

3D Align command for 3D Solid Objects can be found out through following methods.

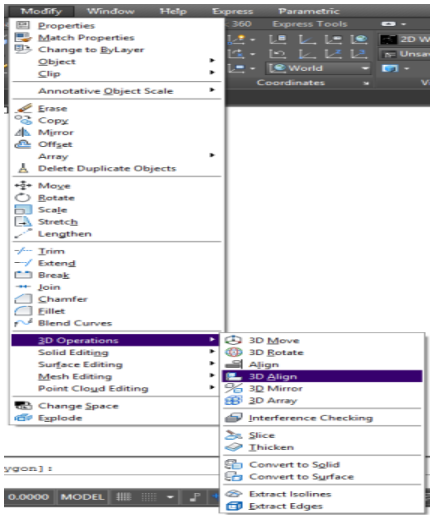
Through Ribbon

This figure describes the location of 3D align command for 3D Solid objects in Home Tab.



Through Main Menu

Main Menu→Modify→3D Operations→ 3D Align

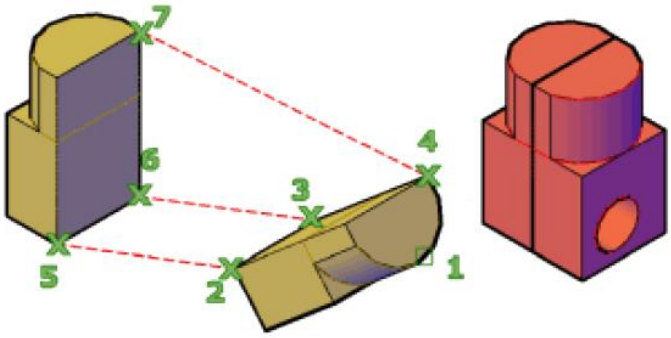


Through Short Cut Key → 3Dalign enter

Application

Type **3Dalign** at the Command Prompt → Select object→ enter →Specify source plane and orientation→ Specify base point or [Copy]: → Specify second point or [Continue] →Specify third point or [Continue] → Specify destination plane and orientation→Specify first destination point→Specify second destination point →Specify third destination point

Note:
If the destination is a plane on an existing solid object, you can define the destination plane with a single point by turning on dynamic UCS.



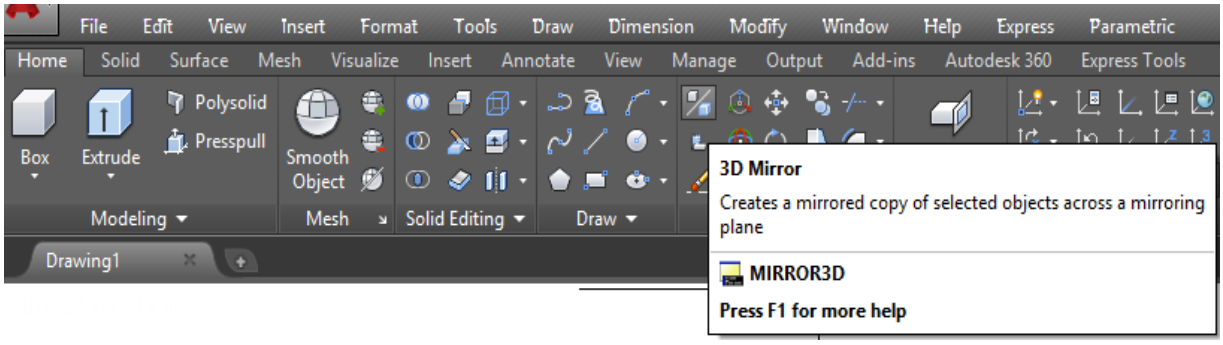
4. 3D Mirror Command

Create a mirrored copy of selected 3D objects across a mirroring plane. You can specify the mirroring plane by aligning the objects with a specified plane or by specifying three points.

3D Mirror command for 3D Solid Objects can be found out through following methods.

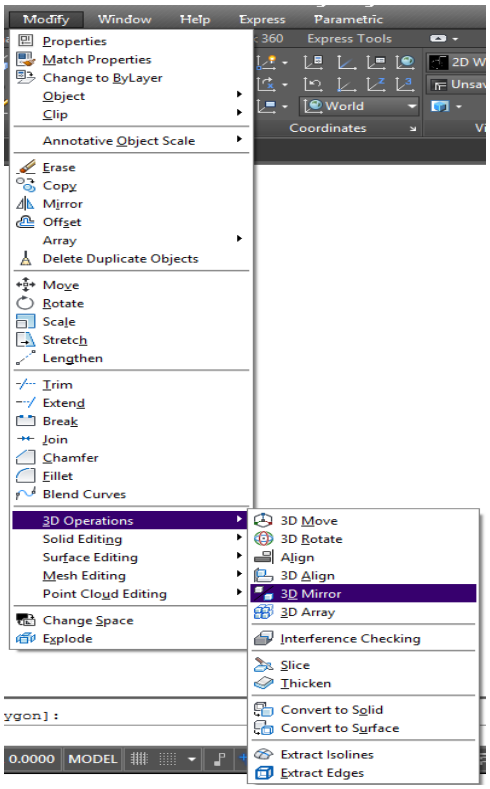
Through Ribbon

This figure describes the location of 3D mirror command for 3D Solid objects in Home Tab.



Through Main Menu

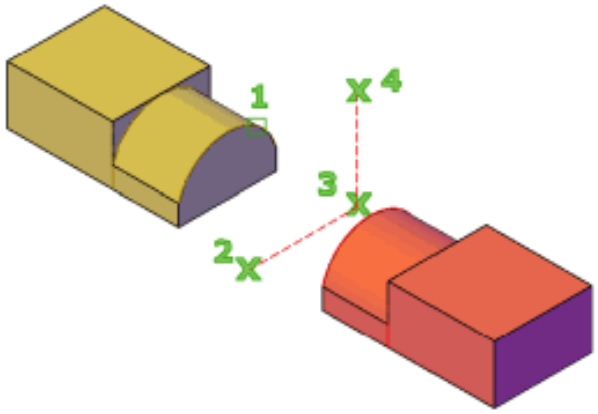
Main Menu→Modify→3D Operations→ 3D Mirror



Through Short Cut Key → 3Dmirror enter

Application

Type **3Dmirror** at the Command Prompt → Select object → enter →Specify first point of mirror plane (3 points) or[Object/Last/Zaxis/View/XY/YZ/ZX/3points] →Specify second point on mirror plane→Specify third point on mirror plane→Delete source objects? [Yes/No] <N>→ enter



5. 3D Array Command

3DARRAY functionality has been replaced with the enhanced ARRAY command, which allows you to create associative or non-associative, 2D or 3D, rectangular, path, or polar arrays. 3DARRAY maintains legacy behavior.

- For 3D rectangular arrays, in addition to columns and rows, you also specify the number of levels in the Z direction.
- For 3D polar arrays, you specify the axis of rotation with any two points in space.

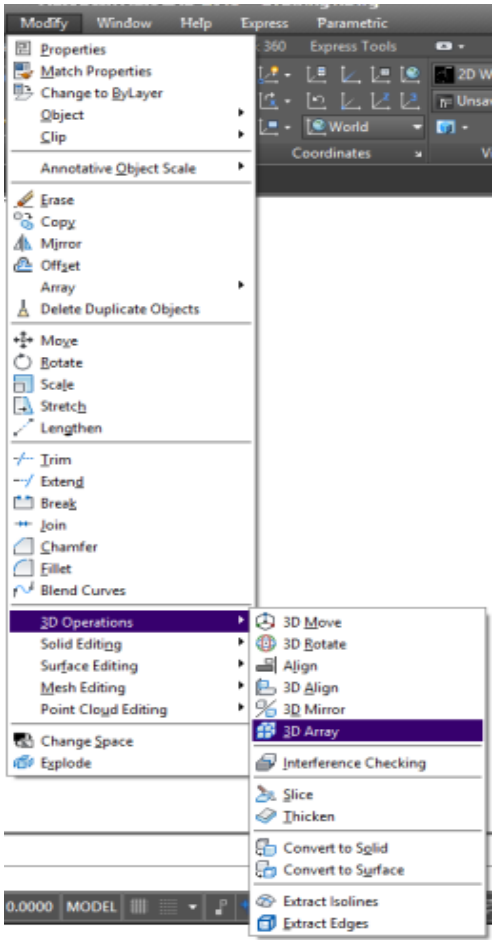
3D Array command for 3D Solid Objects can be found out through following methods.

Through Ribbon

This figure describes the location of 3D Array command for 3D Solid objects in Home Tab.

Through Main Menu

Main Menu→Modify→3D Operations→ 3D Mirror



Through Short Cut Key → 3Darray enter

Array Options

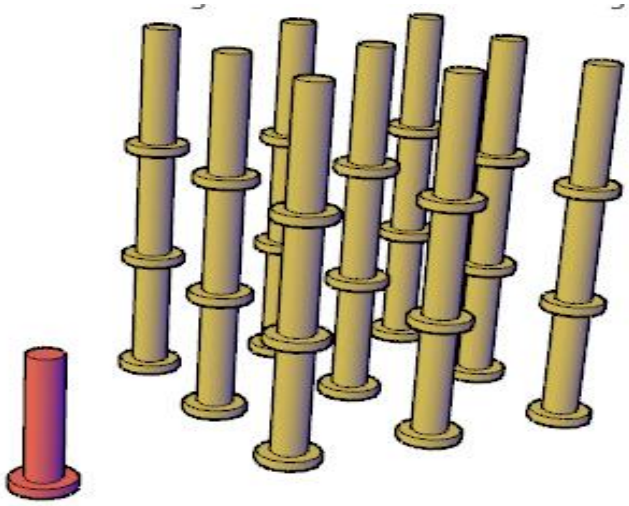
1. 3D Rectangular Array
2. 3D Polar Array

1. 3D Rectangular Array

Copies objects in a matrix of rows (X axis), columns (Y axis), and levels (Z axis). An array must have at least two rows or two columns or two levels. Specifying one row requires that more than one column be specified, and vice versa. Specifying one level creates a two-dimensional array. Positive values generate the array along the positive X, Y, and Z axes. Negative values generate the array along the negative X, Y, and Z axes.

Application

Type **3Dmirror** at the Command Prompt → Select objects→ enter →Enter the type of array [Rectangular/Polar] → R enter(for Rectangular array) →Enter the number of rows (---)→Enter the number of columns (|||)→Enter the number of levels (...)→Specify the distance between rows (---)→Specify the distance between columns (|||)→Specify the distance between levels (...)→enter



Remember

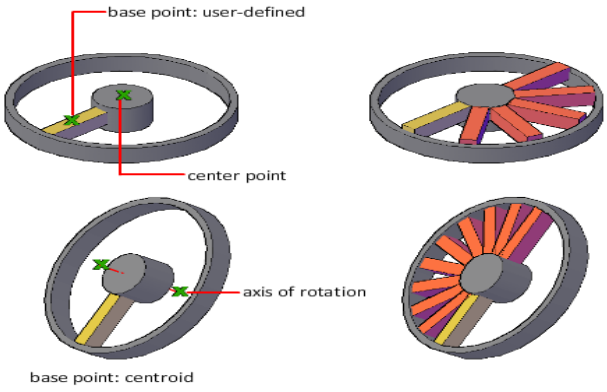
By dragging the array grips, you can increase or decrease the number and spacing of the rows and columns in the array

2. 3D Polar Array

Copies objects about an axis of rotation. The specify angle determines how far the objects are arrayed about the axis of rotation. A positive number produces a counterclockwise array rotation. A negative number produces a clockwise array rotation.

Application

Type **3Dmirror** at the Command Prompt → Select objects→ enter →Enter the type of array [Rectangular /Polar] → P enter (for Polar array) →Enter the number of items in the array→Specify the angle to fill (+=ccw,-=cw) <360> → enter →Rotate arrayed objects? [Yes/No] <Y>: →enter Specify center point of array→Specify second point on axis of rotation→enter



4. Extract Edges Command

Create wireframe geometry from the edges of a 3D solid, surface, mesh, region, or sub object. With the XEDGES command, you can create wireframe geometry by extracting all the edges from the following objects:

- **3D solids**
- **3D solid history sub objects**

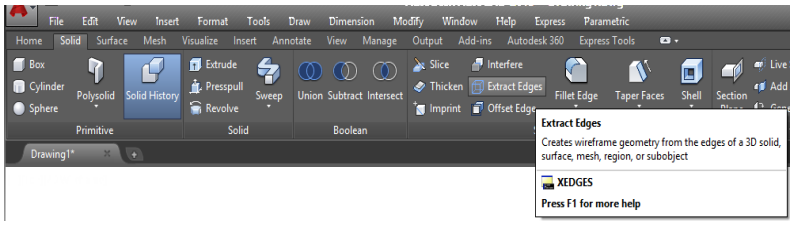
- **Meshes**
- **Regions**
- **Surfaces**
- **Sub objects (edges and faces)**

Press and hold Ctrl to select faces, edges and component objects, repeating if necessary. Objects such as lines, arcs, splines, or 3D polylines are reacted along the edges of the selected objects or sub objects.

Extract Edges command for 3D Solid Objects can be found out through following methods.

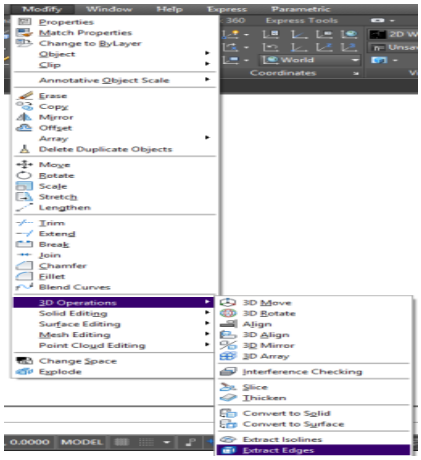
Through Ribbon

This figure describes the location of **Extract Edges** command for 3D Solid objects in Solid Tab.



Through Main Menu

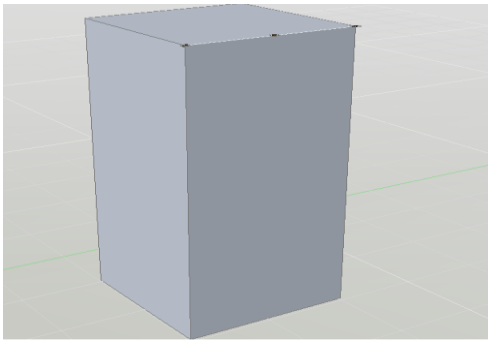
Main Menu→Modify→3D Operations→ **Extract Edges**



Through Short Cut Key → XEDGES enter

Application

Type **XEDGES** at the Command Prompt → Select objects → enter



5. Facetres Command

Adjust the smoothness of shaded and rendered objects and objects with hidden lines removed. By default value of Facetres is 0.5000 and valid values are from 0.01 to 10.0.

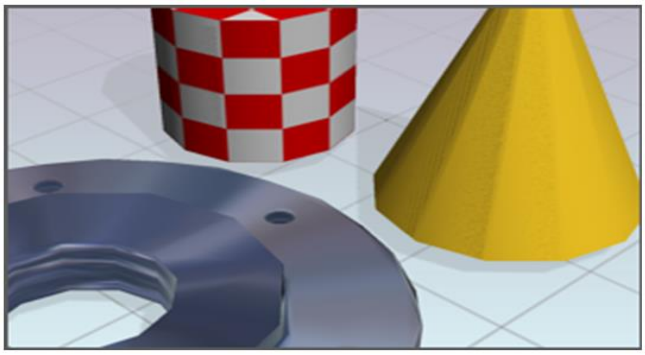
3DDWFPREC replaces FACETRES in controlling the precision of published 3D DWF files. When you use the 3DPRINT command, the FACTRES system variable is set to 10.

Short Cut Key → FACETRES enter

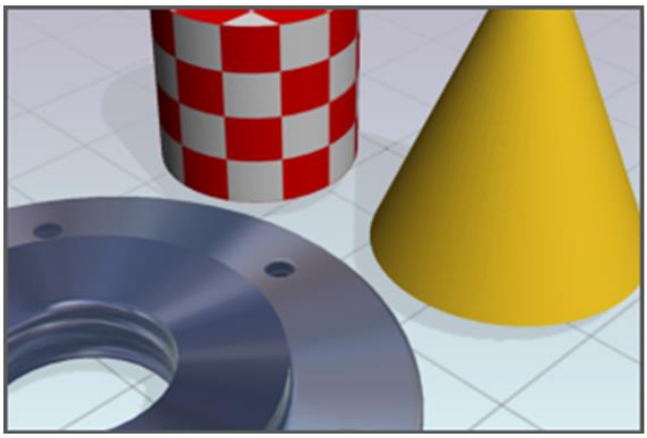
Application

Type **FACETRES** at the Command Prompt → Enter new value for FACETRES <0.25>→put the new value of FACETRES 5→enter

In the following example, facets display on curved geometry when FACETRES is low. FACETRES = .25.



In the following example, smoother geometry is displayed when FACETRES is set to higher values. FACETRES = 5.



6. ISO LINES Command

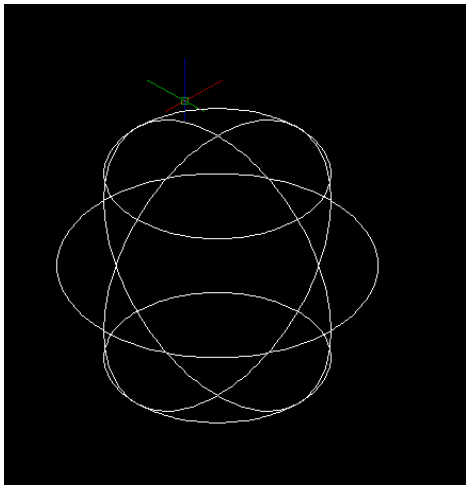
Specify the number of contour lines displayed on the curved surfaces of 3D solids. By default value of ISO LINE is 4.Valid settings range from 0 to 2047.

Short Cut Key → ISOLINES enter

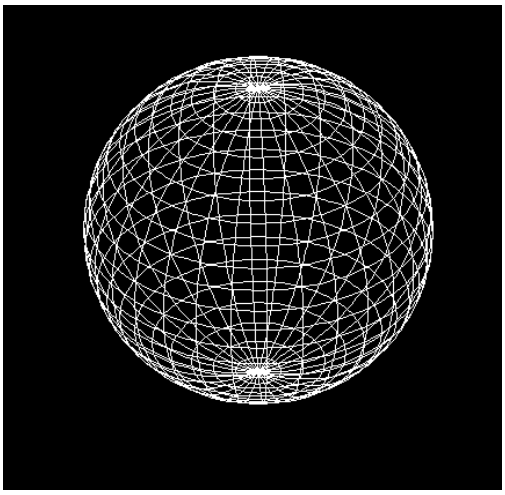
Application

Type **ISOLINES** at the Command Prompt → Enter new value for ISOLINES <4>→put the new value of ISOLINES 20 →enter.

In the following example, facets are displayed on curved geometry when ISOLINES is low. ISOLINES = 04



In the following example, smoother geometry is displayed when ISOLINES is set to higher values. ISOLINES = 30.



Learning Unit 3: Develop 3D Solid Composites

Overview

In this unit, you will learn about the development of 3D solid composites. You can combine, subtract or intersect two or more 3D solids by using Boolean operating, able to adjust the thickness of 3D solid, you can draw a temporary 3D solid by interesting two sets of 3D solids.

1. Boolean Operations

You can use the Boolean operations to create complex solid models out of primitives. The following tutorial shows you how to use Intersect, Subtract, and Union operations

Composite functions on solids:

Remember

You cannot use UNION with mesh objects. However, if you select a mesh object, you will be prompted to convert it to a 3D solid or surface.

1. Union
2. Subtract
3. Intersect

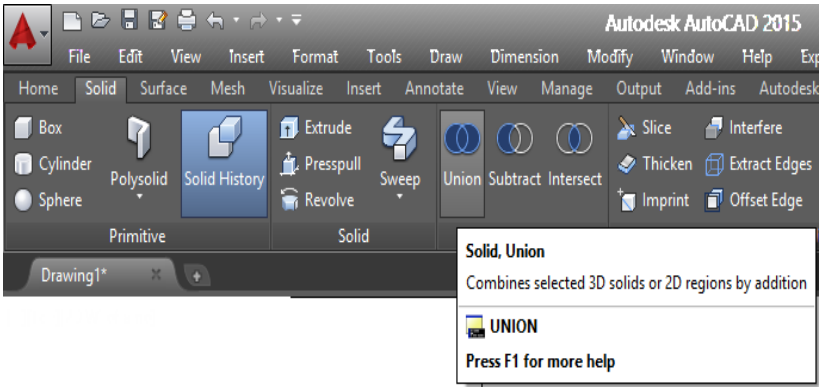
1. Union

You can combine two or more 3D solids, surfaces, or 2D regions into a single, composite 3D solid, surface, or region. You must select the same type of objects to combine.

Union command for 3D Solid Objects can be found out through following methods.

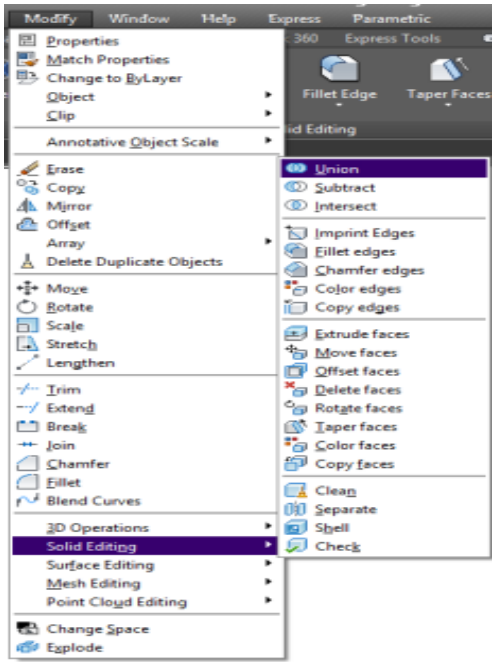
Through Ribbon

This figure describes the location of **Union** command for 3D Solid objects in Solid Tab.



Through Main Menu

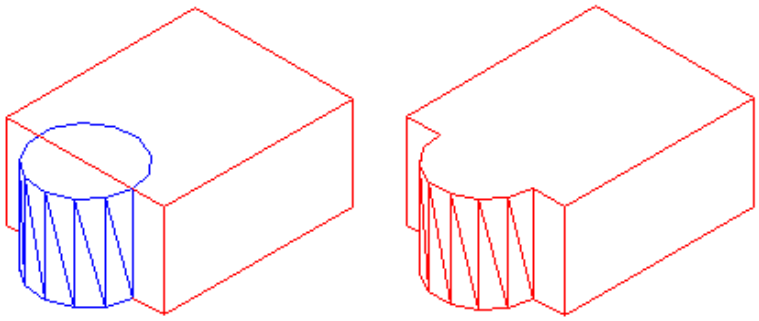
Main Menu→Modify→Solid Editing→ **Union**



Through Short Cut Key → uni enter

Application

Type **UNI** at the Command Prompt →Select objects→ enter



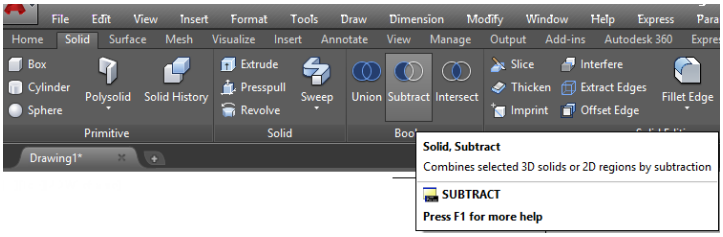
2. Subtract

With SUBTRACT, you can create a 3D solid by subtracting one set of existing 3D solids from another, overlapping set. You can create a 2D region object by subtracting one set of existing region objects from another, overlapping set. You can select only regions for use with this command.

Subtract command for 3D Solid Objects can be found out through following methods.

Through Ribbon

This figure describes the location of **Subtract** command for 3D Solid objects in Solid Tab.

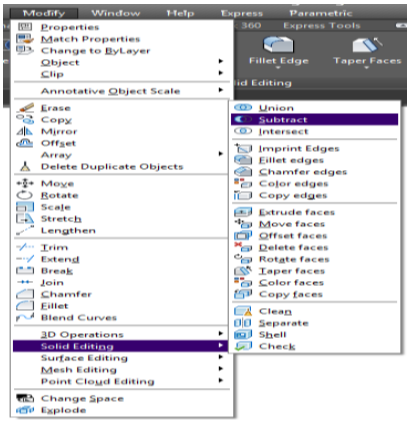


Through Main Menu

Main Menu→Modify→Solid Editing→ **Subtract**

Remember

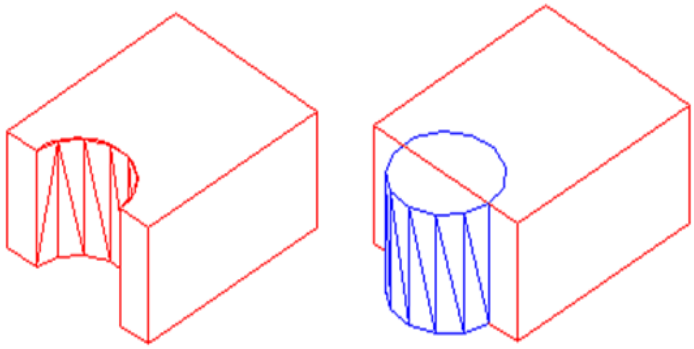
Using SUBTRACT with 3D surfaces is not recommended.



Through Short Cut Key → su enter

Application

Type **SU** at the Command Prompt → SUBTRACT Select solids and regions to subtract from→enter→ Select solids and regions to subtract → enter



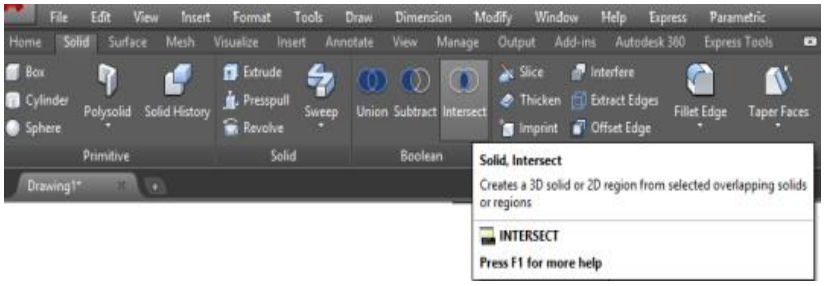
3. Intersect

Create a 3D solid, surface, or 2D region from overlapping solids, surfaces, or regions. With INTERSECT, you can create a 3D solid from the common volume of two or more existing 3D solids, surfaces, or regions. If you select a mesh, you can convert it to a solid or surface before completing the operation.

Intersect command for 3D Solid Objects can be found out through following methods.

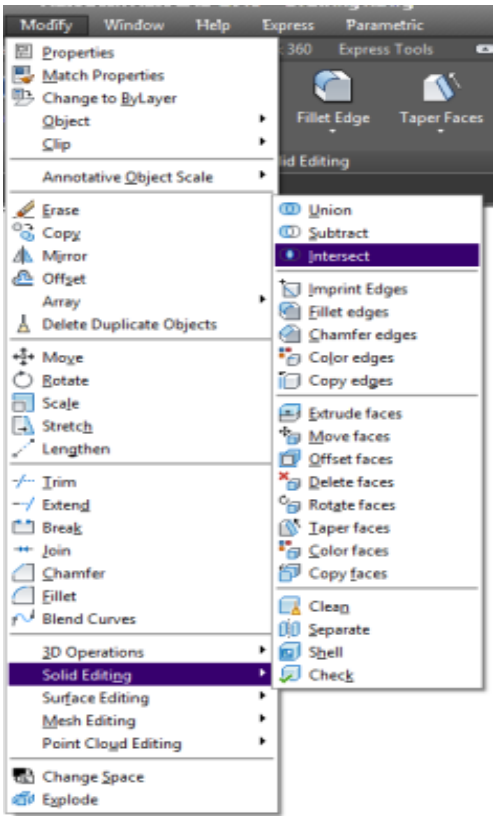
Through Ribbon

This figure describes the location of **Intersect** command for 3D Solid objects in Solid Tab.



Through Main Menu

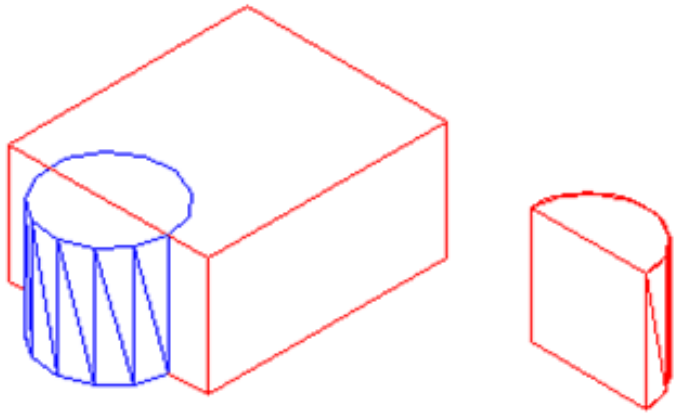
Main Menu→Modify→Solid Editing→ **Intersect**



Through Short Cut Key → in enter

Application

Type **IN** at the Command Prompt → Select objects→ **enter**



2. Thicken Command

Convert a surface into a 3D solid with a specified thickness. A useful technique for modeling a complex 3D curved solid is to first create a surface and then convert it to 3D solid by thickening it. Initially, the default thickness value is zero. During a drawing session, the default value for the thickness is the previously entered thickness value.

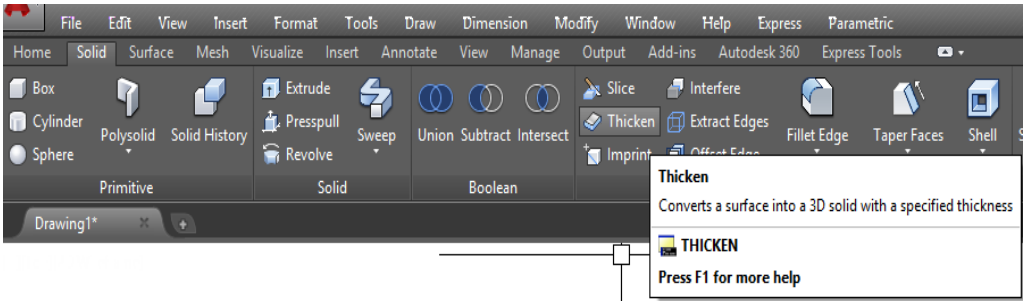
Thicken command for 3D Solid Objects can be found out through following methods.

Through Ribbon

This figure describes the location of **Thicken** command for 3D Solid objects in Solid Tab.

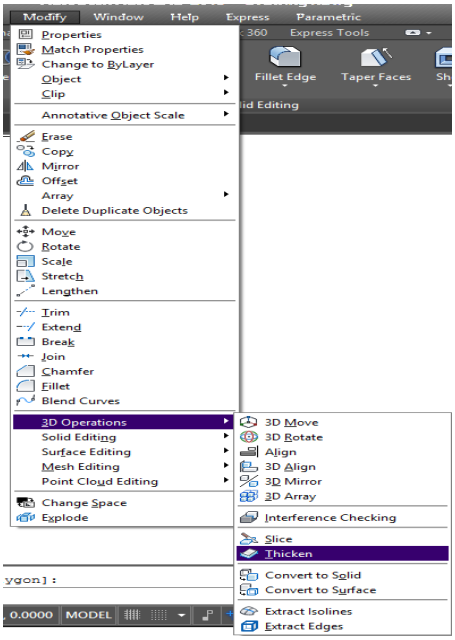
Remember

If you select a mesh face to thicken, you can choose to convert the mesh object
To a solid or surface before completing the operation.



Through Main Menu

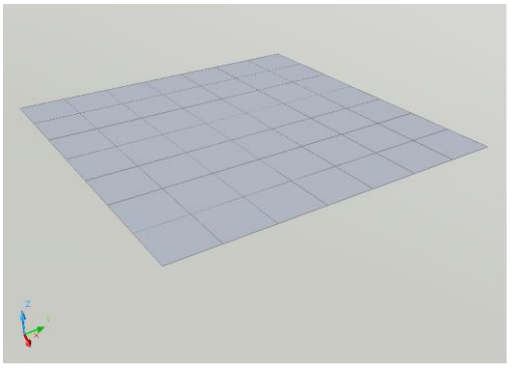
Main Menu→Modify→3D operations → **Thicken**

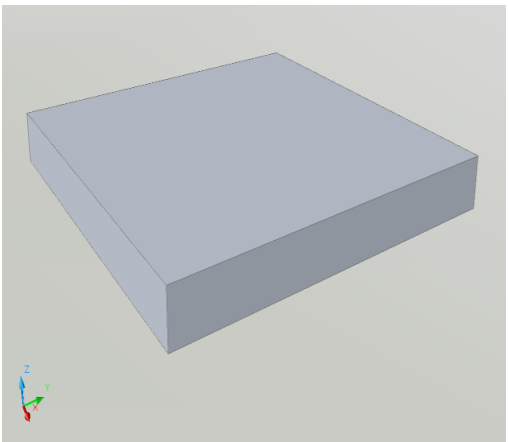


Through Short Cut Key → thicken enter

Application

Type **THICKEN** at the Command Prompt → Select surfaces to thicken→Specify thickness <0.0000>→put the thicken value→enter





3. Interfere Command

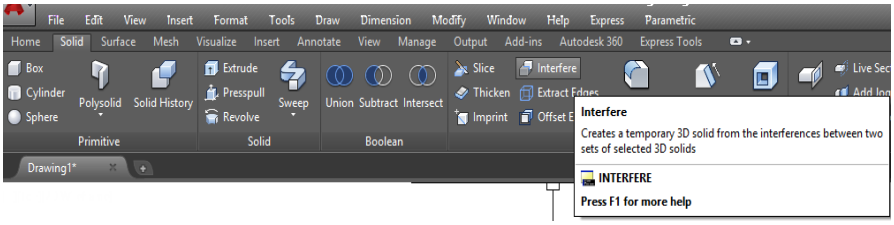
Create a temporary 3D solid from the interferences between two sets of selected 3D solids. Interferences are highlighted with a temporary 3D solid that represents the intersecting volume. You can also choose to retain the overlapping volumes

INTERFERE highlights all interfering 3D solids and displays the number of objects selected and the number of interfering pairs. If you define a single selection set, INTERFERE checks all the solids in the set against one another. If you define two selection sets, INTERFERE checks the solids in the first selection set against those in the second selection set. If you include the same 3D solids in both selection sets, INTERFERE considers the 3D solid part of the first selection set and ignores it in the second selection set.

Interfere command for 3D Solid Objects can be found out through following methods.

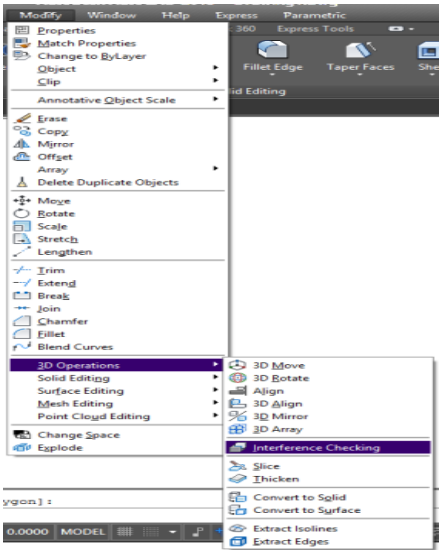
Through Ribbon

This figure describes the location of **Interfere** command for 3D Solid objects in Solid Tab.



Through Main Menu

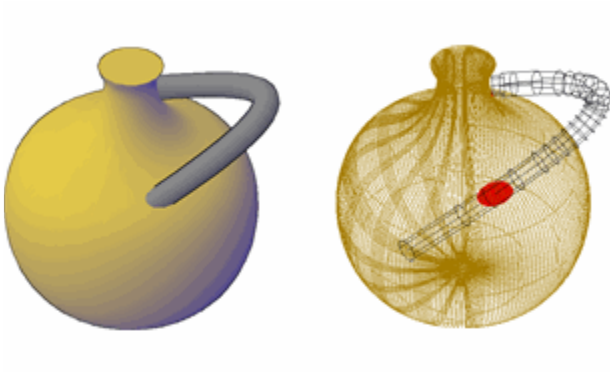
Main Menu→Modify→3D operations → **Interfere**



Through Short Cut Key → inf enter

Application

Type **INF** at the Command Prompt → Select first set of objects or [Nested selection/Settings] →Select solids →enter.



Learner's Activity

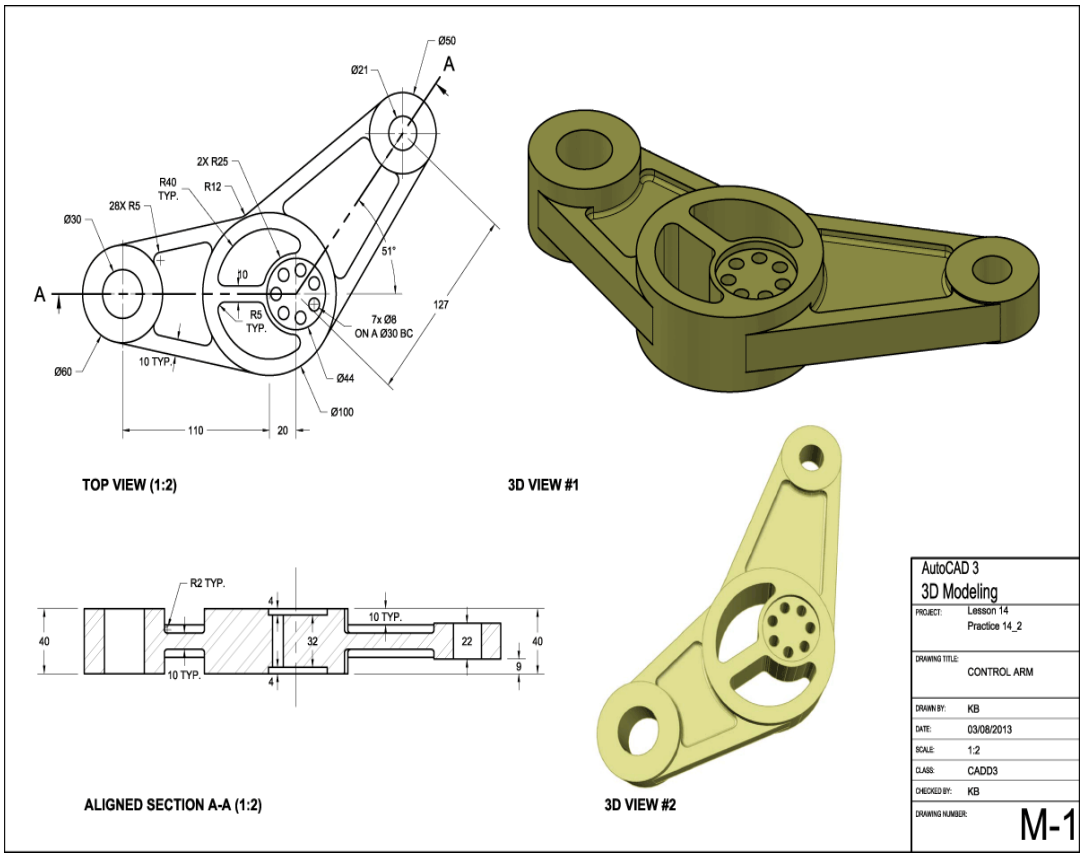
Project # 1

1. Draw a House Plan according to following requirements
   (For Civil Department)

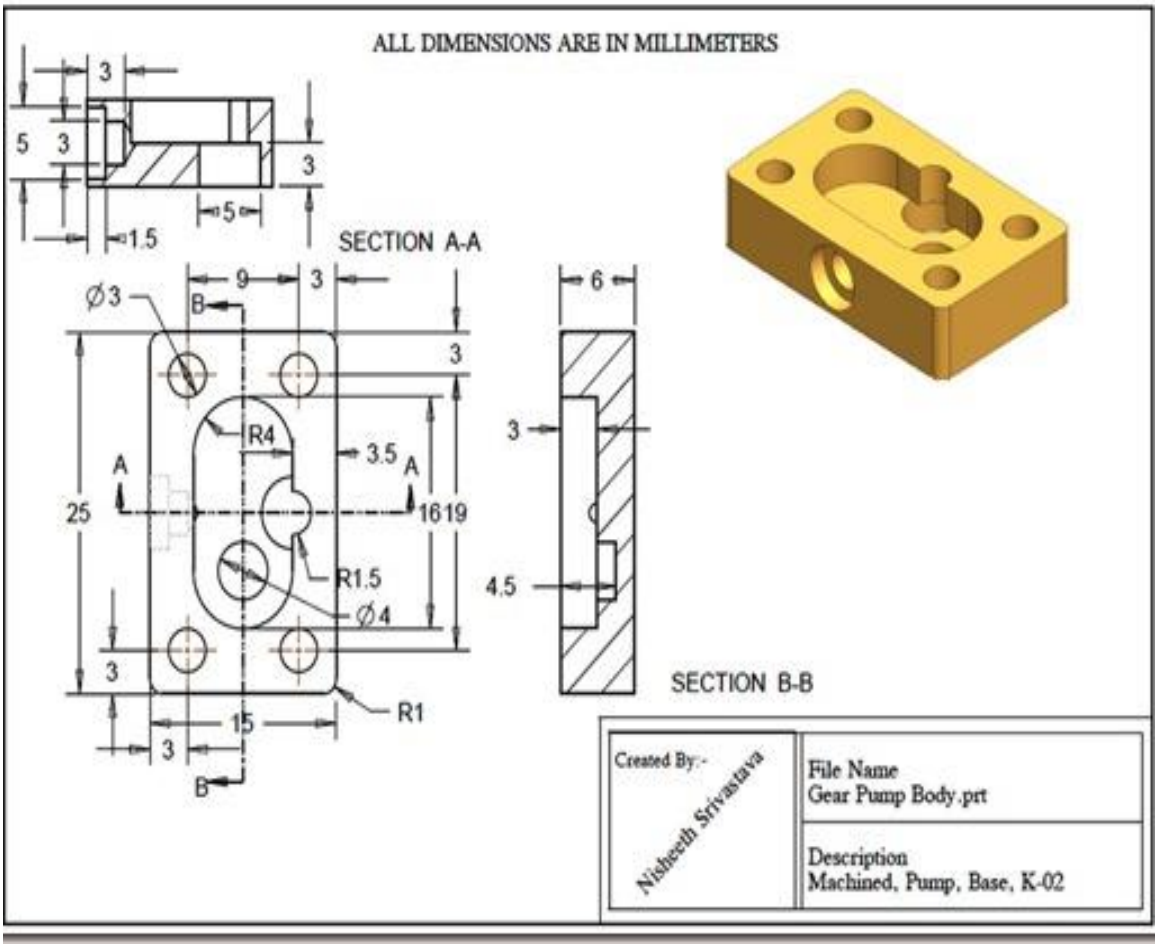
   A. **Plot Size 45'X60'(Regular)**
   B. **Requirements**
      - **Two Bed with Attached bath**
      - **Kitchen**
      - **Drawing Room**
      - **Dinning**
      - **T. V Launch**
      - **Car porch**
      - **Store**

Draw the given projects according to requirements as mention by Using Solid and Solid edit commands

Project # 2



Project # 3


ALL DIMENSIONS ARE IN MILLIMETERS
SECTION A-A
SECTION B-B
Created By:-
Nisheeth Srivastava
File Name
Gear Pump Body.prt
Description
Machined, Pump, Base, K-02

Summary of Module

- In this unit we learnt How to create 3D solid objects. Such as boxes, cones cylinder, spheres, wedges, pyramids and tore, we learnt to draw cross solids by joining or subtracting two or more solids, how to convert a line into poly line, able to extrude open / closed objects to draw 3D surface or solid. Beside primary commands such as extrude and revolve advance commands like sweep commands used to draw a solid by sweeping open / closed, Planer /non planer curve along open /closed path, left command used to draw a solid by joining different cross section along any axis.

- We also learnt different commands for editing a 3D solid object, how to convert 3D meshes polylines and circles to 3D solid, able to adjust the smoothness level of resulting 3D object. We learnt to create a surface from 2D/3D solid , lines, area or mesh objects, learnt about the application of solid editing commands such as 3D more , rotate , align , and creation of mirror coping of selected 3D objects. We learnt about the development of 3D composites, about Boolean operation to combine, subtract or intersect to or more 3D solid, finally we learnt about the adjustment of thickness and smoothness of resulting 3D objects, and to draw a 3D solid by interfering to sets of 3D solids.

Frequently Asked Questions (FAQs)

FAQ 1: How many type of Solid Primitive command are there?

Answer

There are seven type of solid Primitive command.

1) Box
2) Cone
3) Cylinder
4) Pyramid
5) Sphere
6) Wedge
7) Tours

FAQ 2: Use of Polysolid Command?

Answer

You can create walls with straight and curved segments of constant height and width. With the POLYSOLID command,

FAQ 3: Define Polysolid Options?

Answer

Use the following options to control the size and shape of the polysolid you create

i. Object
ii. Height
iii. Width
iv. Justify

i. Object
Specify an object to convert to a solid. You can convert:
- Line
- Arc
- 2D polyline
- Circle

ii. Height
Specify the height of the solid

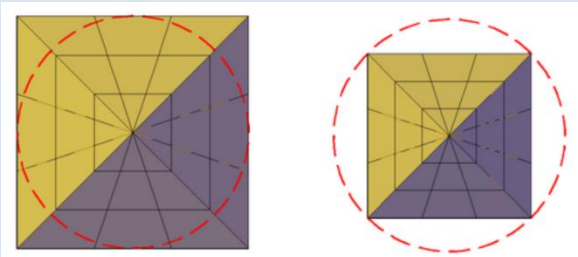
iii. Width
Specify the width of the solid.

iv. Justify
Set the width and height as the solid to be left, right, or center justified when defining the profile with the command.

FAQ 4: Define inscribed or circumscribed perimeter?

Answer

Specify whether the pyramid base is drawn inside or outside of the radius.



Circumscribed Inscribed

FAQ 5:	How can create a cone Frustum?
Answer	Use the Top Radius option to create a cone frustum
FAQ 6:	Name of Different Torus Options?
Answer	The Torus option provides several methods for Determining the size and rotation of the mesh tours you create. i. 3P ii. 2p iii. TTR iv. Radius
FAQ 7:	How can you select face of 3D object?
Answer	Press Ctrl+Click to select faces and edges on 3D objects.
FAQ 8:	What is the use of loft command?
Answer	LOFT command creates a solid by joining several crass sections lying on different planes.
FAQ 9:	How can you create opposite copy of object?
Answer	Create a mirrored copy of selected 3D objects across a mirroring plane. You can specify the mirroring plane by aligning the objects with a specified plane or by specifying three points.
FAQ 10:	How can you find intersect command through main menu?
Answer	Main Menu→Modify→Solid Editing→ Intersect

Test Yourself!

Please mark the correct one from the given options. You can check your answer with the Answer Key at the end of this module.

1. What is the valid arrange of ISOLINES?

a. 0-2056 b. 0-2047

c. 0-2067 d. 0-2058

2. What is the value of FACETRES by Default?

a. 0.500 b. 0.100

c. 0.200 d. 0.25

3. How many types of array?

a. 3 b. 4

c. 2 d. 5

4. TTR stand for _____.

a. Tangent, Tangent, Radius b. Radius ,Tangent, Tangen,

c. ,Tangent, Radius ,Tangent, d. None of all

5. Shortcut of Extrude Command?

a. E b. Ex

c. Extr d. Ext

Answers Key

MCQ Number	Correct Answer
1	b
2	a
3	c
4	a
5	d

AUTOCAD

Module-9

LEARNING GUIDE

National Vocational Certificate Level 3

Version 1 - July 2013

Module 9: Modify Solid Faces

Learning Outcomes

After completion of this learning module, you will be able to:

- Modify Solid Faces
- Edit solid Objects

Learning Unit 1: Modify Solid Faces

Overview

This unit describes how to modify 3D solid object faces. You will learn about the application of different commands to modify 3D solid objects. You will be able to Extrude, Copy, and Delete, Color faces, Imprint and Execute shell effects from 3D solid objects.

Remember

This sign is for moving to next step.

1. Modify Solid Faces

You can modify faces of 3D solids by extruding, Tapering, deleting, copying or changing the color of the selected faces.

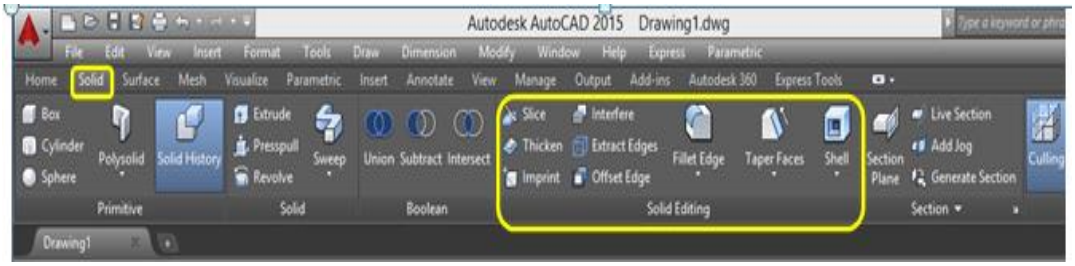
Location on Solid Editing Tools

There are two methods for finding the location of solid editing tools

1. Through Ribbon
2. Through Main Manu

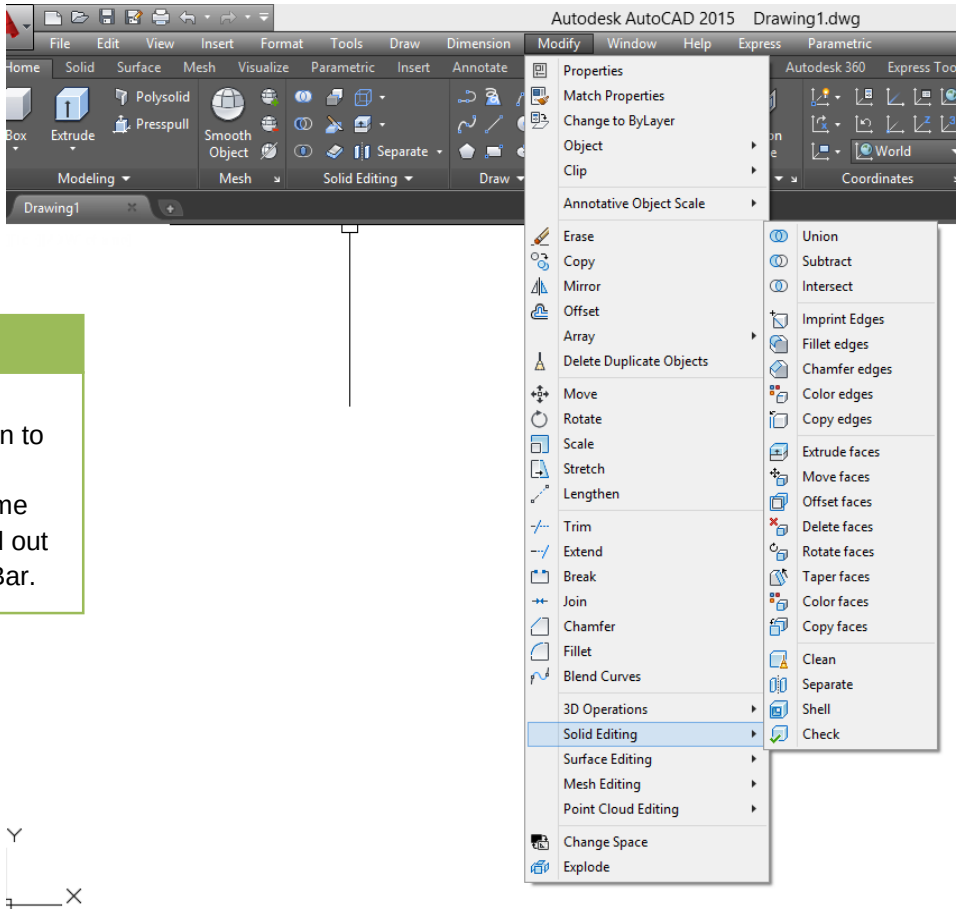
1) Through Ribbon

Figure shows Solid Editing Through Ribbon.



2) Through Menu Bar

Menu Bar→Modify →Solid Editing



Remember

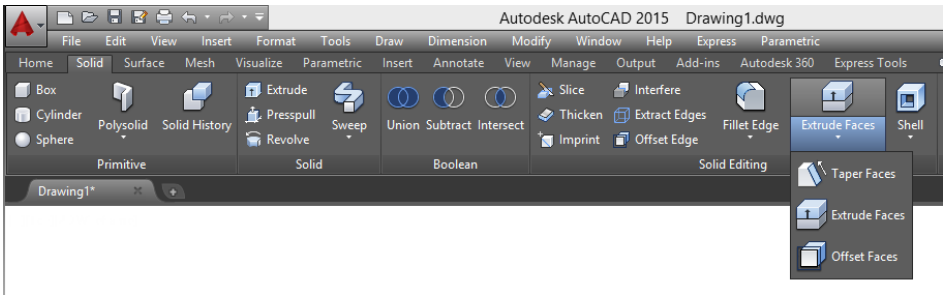
Some tools are present in ribbon to modify 3D solid object while some tools will be find out through Menu Bar.

Extrude Faces of 3D Solid

You can extrude one or more faces of 3D solid at the specified height and path. Extrude faces command of 3D solid can be find out through both methods.

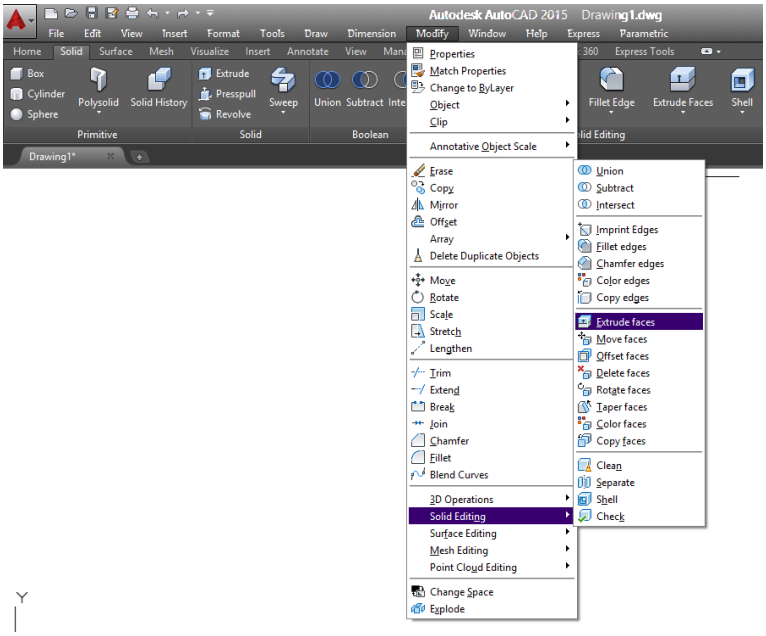
Through Ribbon

This figure describes the location of extrude faces of 3D solid in solid tab.



Through Menu Bar

Menu Bar→Modify →Solid Editing→Extrude faces



There are two options of Extrude faces command

1. Height
2. Path

1) APPLICATION (Extrude faces with Height)

Pick tool → select faces → put the Height of extrusion→put the taper angle of extrusion

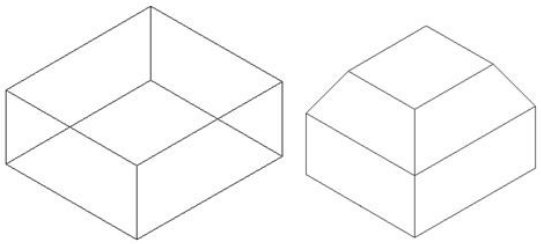


Figure 1

Figure 2

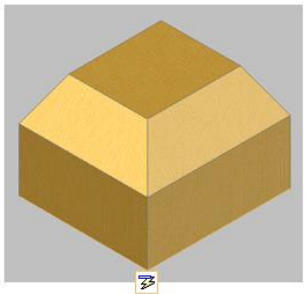
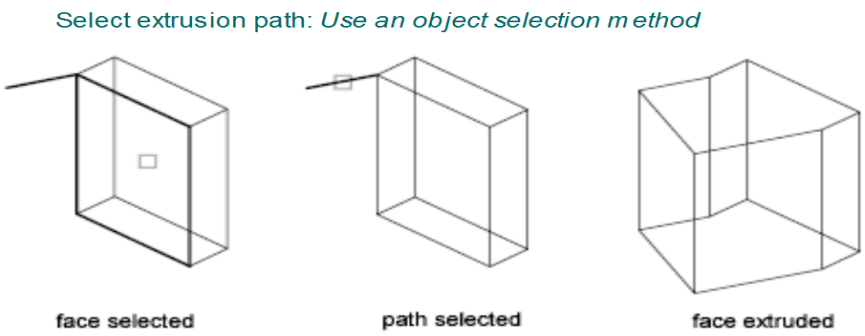


Figure 3

APPLICATION (Extrude faces with Path)

In the first step we draw the path in polyline.

Pick tool → select faces →put the “P” for path option → Select the path



Move Faces of 3D Solid

This command Moves the selected face on a 3D solid object to a specified height or distance.

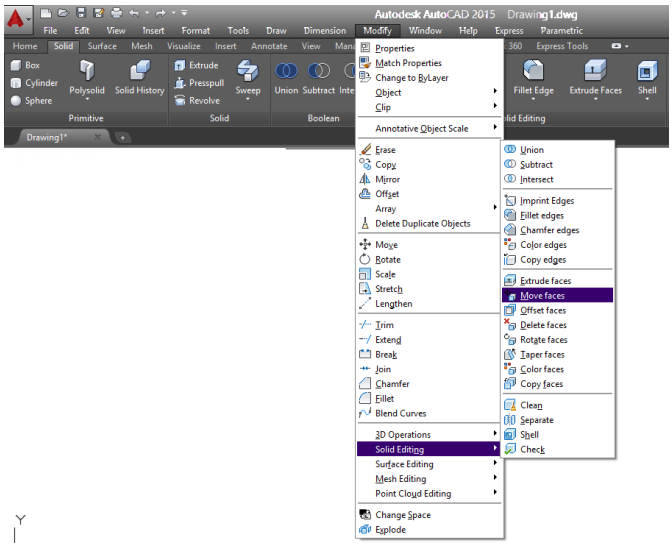
Move faces command of 3D solid can be find out only through Menu Bar.

Through Main Manu

Menu Bar→Modify →Solid Editing →Move faces

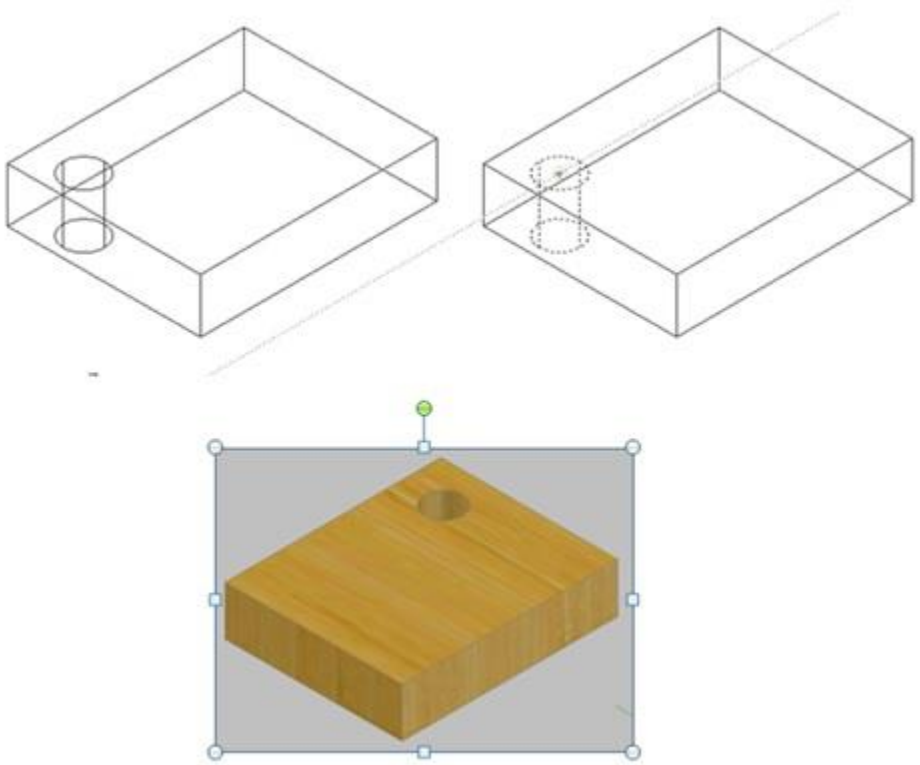
Remember

You can also move multiple faces at one time using move faces command.



APPLICATION

Pick tool → select faces →select base point→put the value of displacement

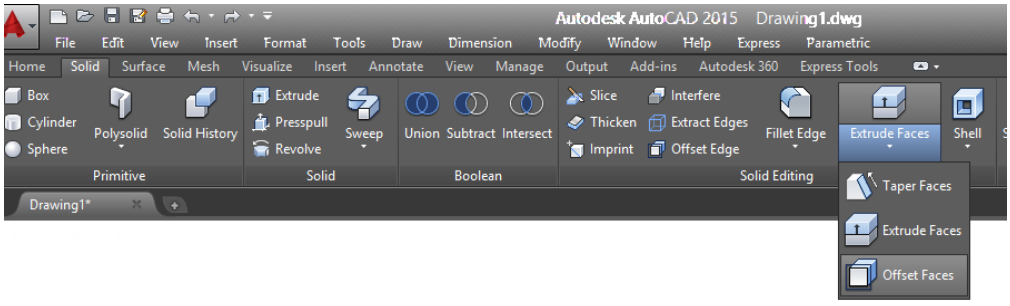


Offset Faces of 3D Solid

Offset faces command of 3D solid is used to increase/decrease volume up to a specified distance. A positive value increases the size or volume of the solid where as negative value decreases the size or volume of the solid.

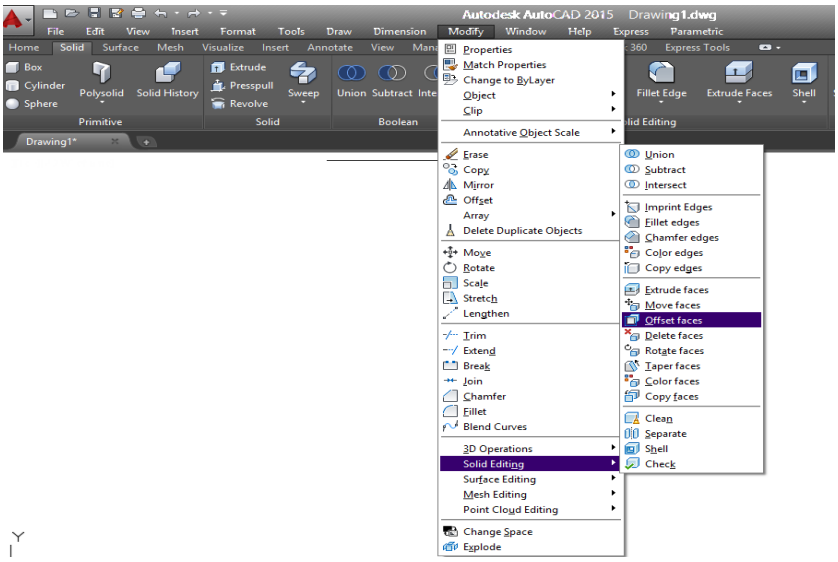
Offset faces command of 3D solid can be find out through both methods.

Through Ribbon



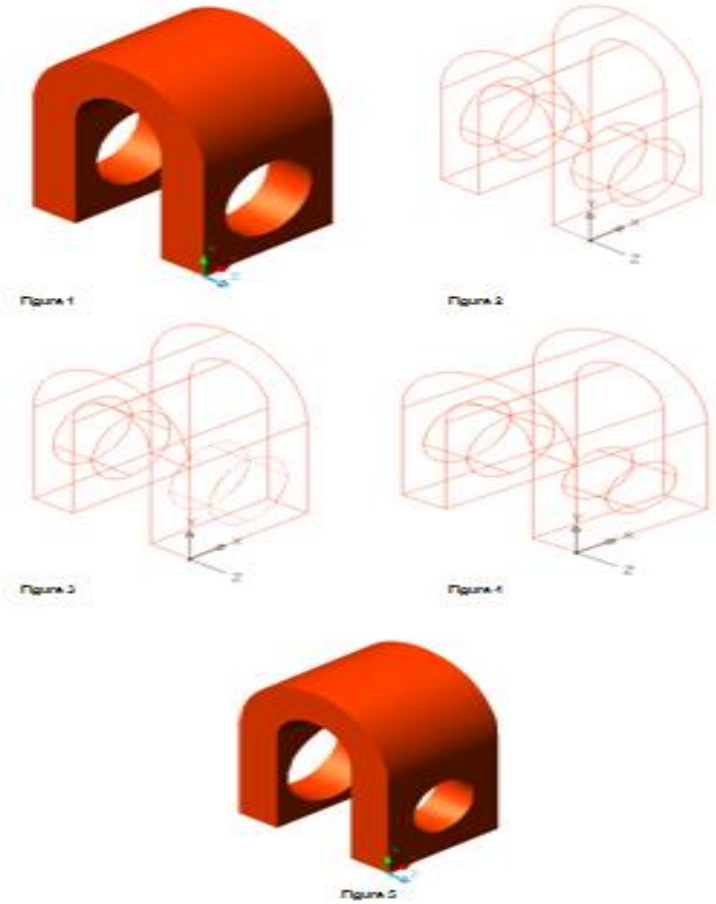
Through Menu Bar

Menu Bar→Modify →Solid Editing →offset faces



APPLICATION

Pick tool → select face→ enter →put the offset distance →enter



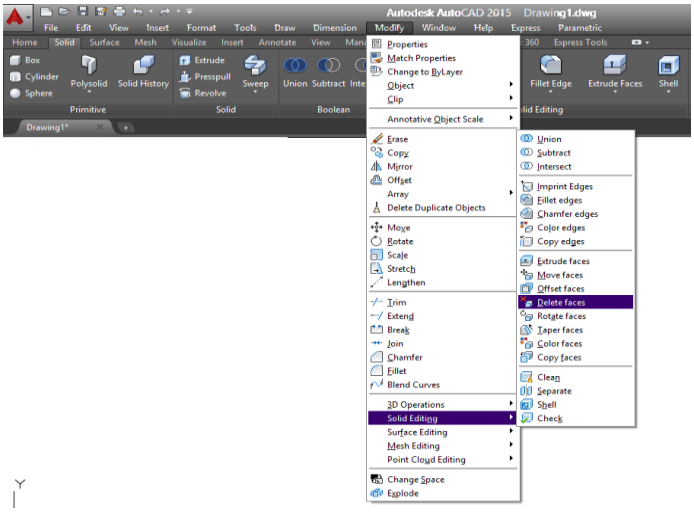
Delete Faces of 3D Solid

Delete faces command of 3D solid enables us to delete or remove faces, including fillets and chamfers.

Delete faces command of 3D solid can be find out only through Menu Bar.

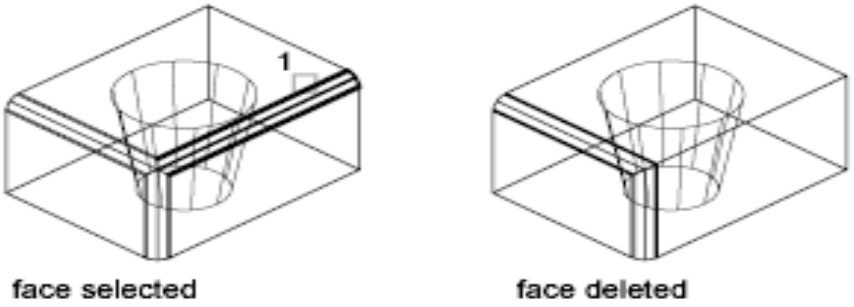
Through Main Manu

Manu →Modify →Solid Editing →Delete faces



APPLICATION

Pick tool → select faces →enter



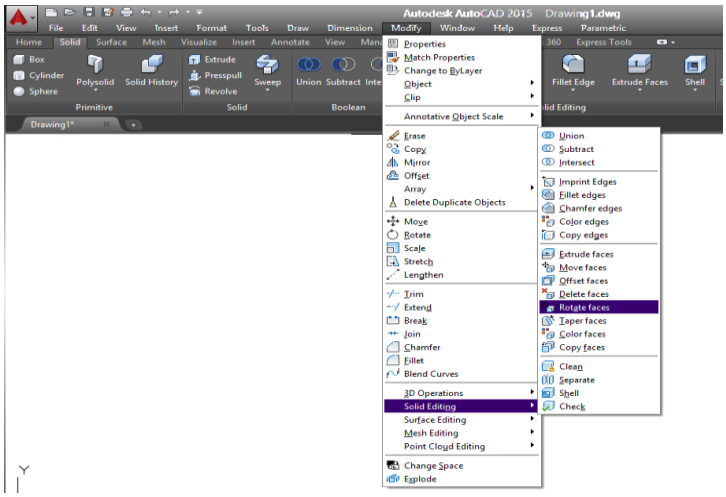
Rotate Faces of 3D Solid

Rotate faces command of 3D solid is used to rotate one or more faces or a collection of features on a solid about a specified axis.

Rotate faces command of 3D solid can be find out only through Menu Bar.

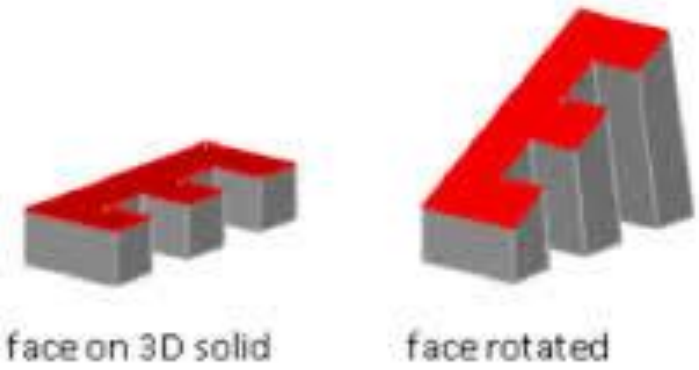
Through Menu Bar.

Menu Bar→Modify →Solid Editing →Rotate faces



APPLICATION

pick tool → specify select face→ enter →put the offset distance →enter

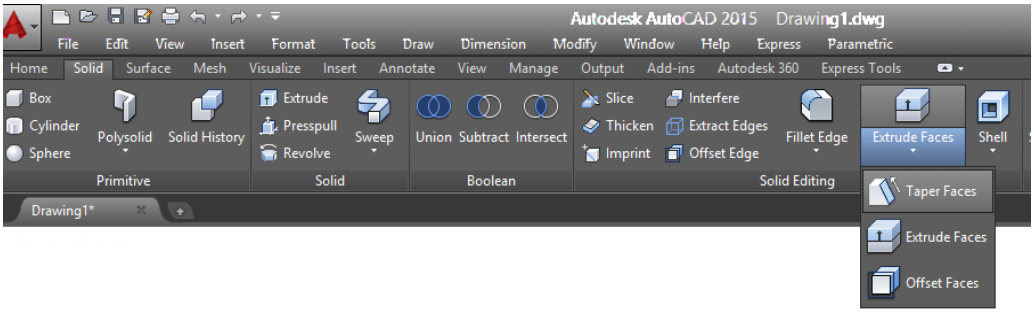


Taper Faces of 3D Solid

Taper faces command of a 3D solid is used to taper faces at specified angle. The rotation of the taper angle is determined by the selection sequence of the base point and second point along the selected vector.

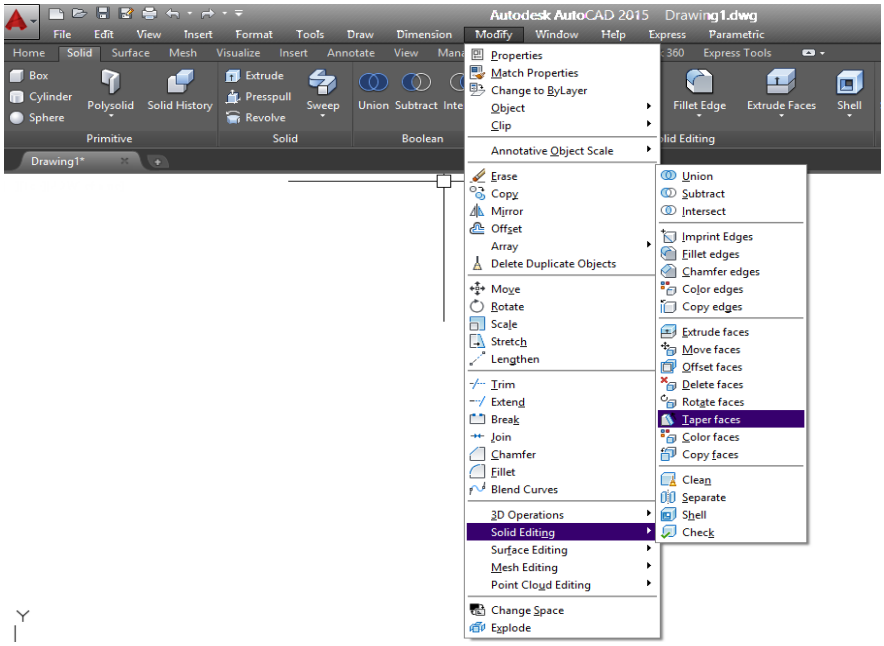
Tapered faces command of 3D solid can be find out through both methods.

Through Ribbon



Through Main Manu

Menu Bar→Modify →Solid Editing →Taper faces



APPLICATION

Pick tool → select faces →select base point →other point along the axis →put the taper angle

.

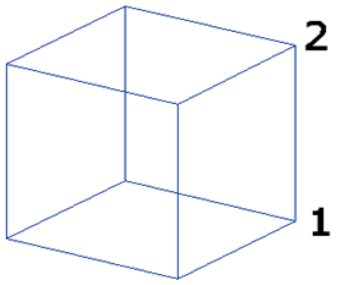


Figure 1

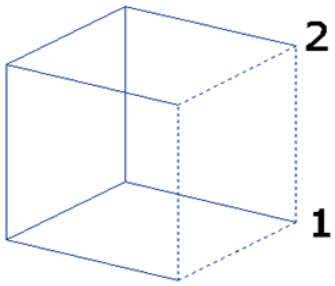


Figure 2

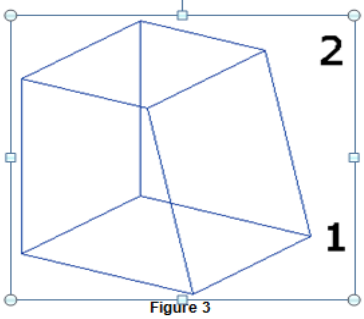


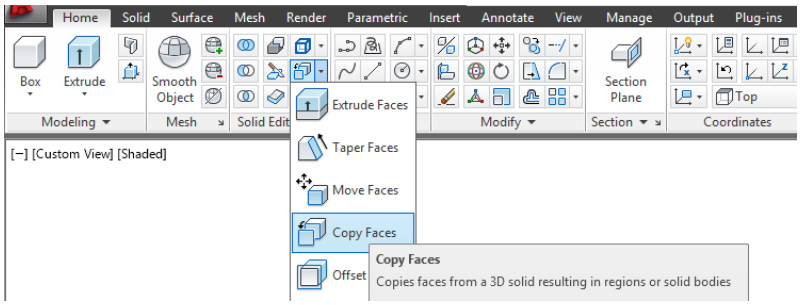
Figure 3

Copy Faces of 3D Solid

Copy faces command of 3D solid is used to copy faces as a region or a body. If you specify two points, SOLIDEDIT uses the first point as a base point and places a single copy relative to the base point.

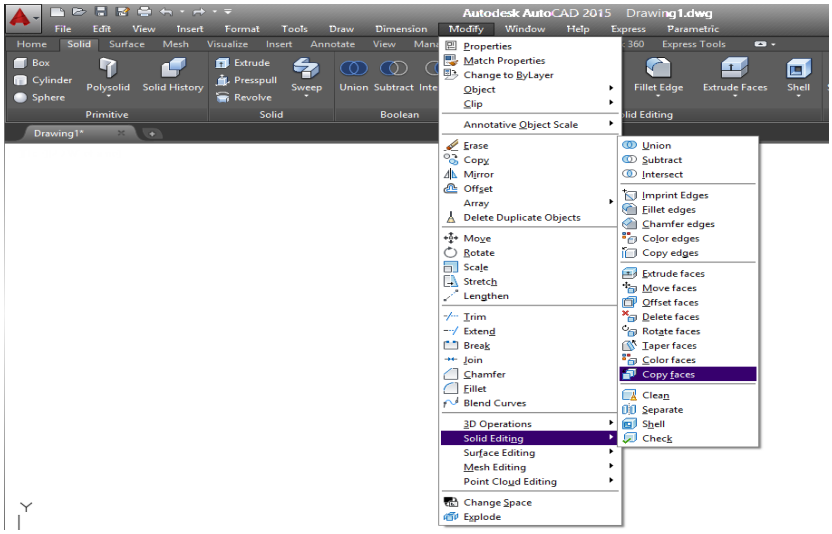
Copy faces of 3D solids can be find out only through both methods.

Through Ribbon



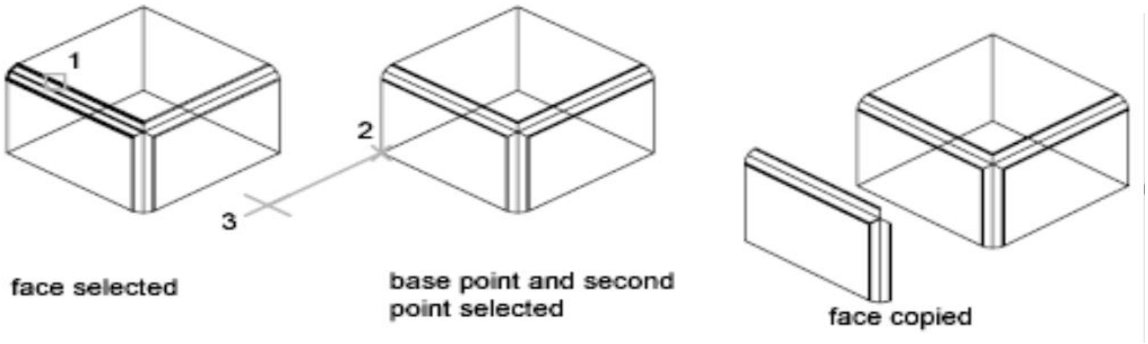
Through Menu Bar

Menu Bar→Modify →Solid Editing →copy faces



APPLICATION

Pick tool → select faces →click Base point→put the distance of placement→Enter

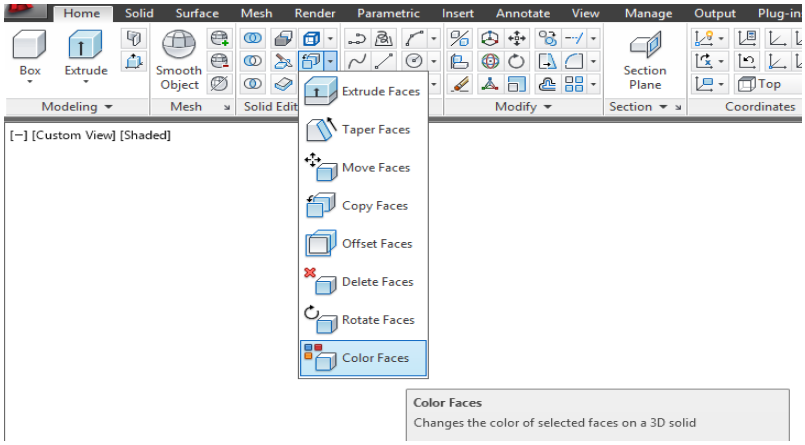


Color Faces of 3D Solid

You can also assign color to the faces of 3D solid by using Color faces of 3D solid command. The color you selected through this command overwrites the color of the layer on which the faces are drawn.

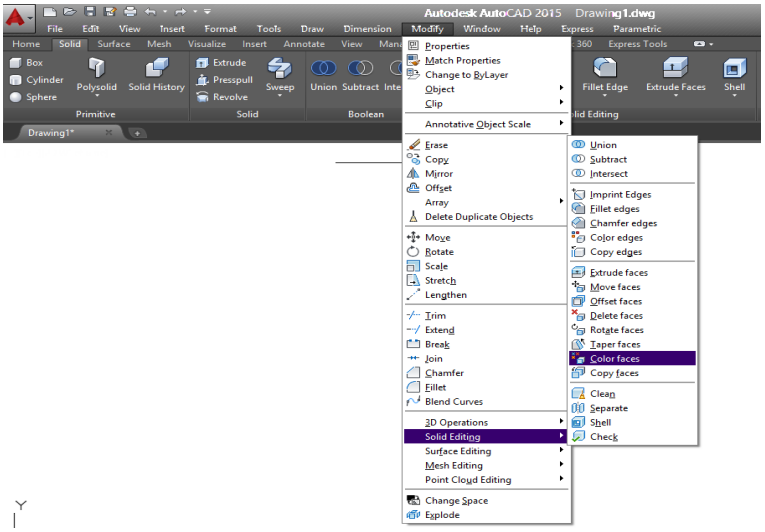
Copy faces of 3D solids can be find out only through both methods.

Through Ribbon



Through Menu Bar

Menu Bar→Modify →Solid Editing →color faces



APPLICATION

Pick tool → select faces →enter →select the color→ok

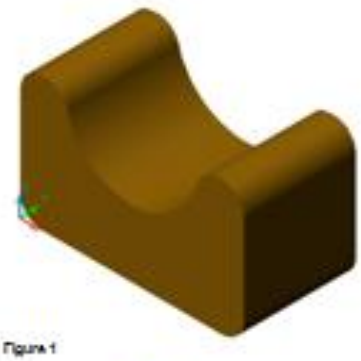


Figure 1

Figure 2

Figure 3

Figure 4

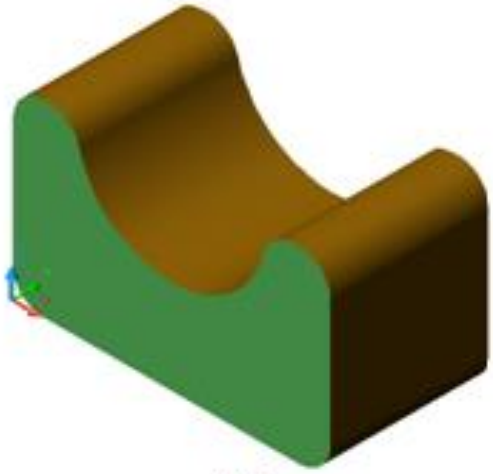


Figure 5

How to Apply “Imprint” Facility on 3D Solid Object

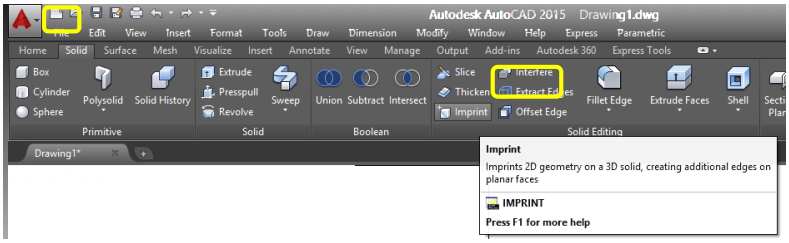
Do you know!

Imprint means to draw a 2D geometry on a 3D solid faces by creating additional edges in plane

This command is used to imprint an object on the selected solid face. The object to be imprinted must intersect one or more faces on the selected solid in order for imprinting to be successful. Imprinting is limited to the following objects: arcs, circles, lines, 2D and 3D polylines, ellipses, sp lines, regions, bodies, and 3D solids.

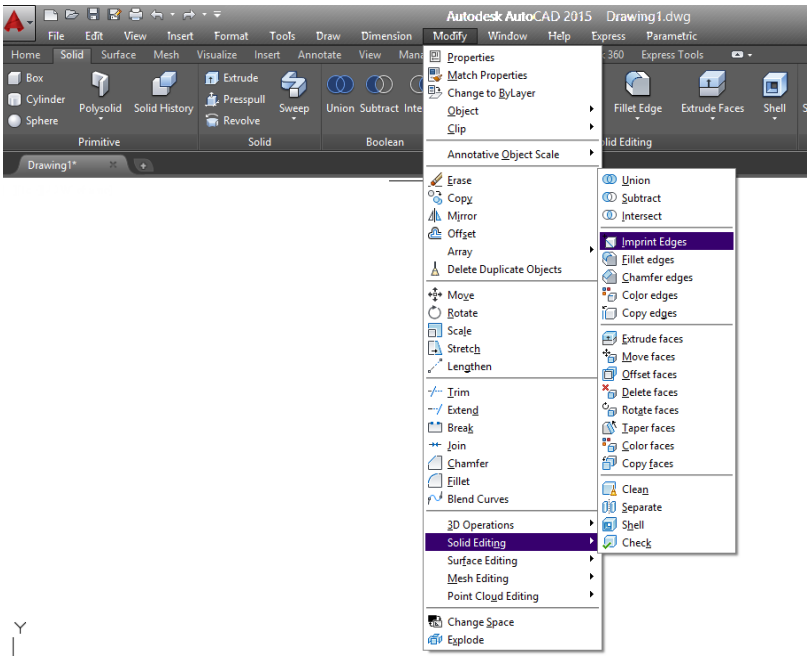
Imprint command can be find out through both methods.

Through Ribbon



Through Main Manu

Menu Bar→Modify →Solid Editing →imprint edges



APPLICATION

Draw the shape in 2D polylines for imprinting on 3D solid face.

Pick tool → Select 3D object →Select imprint object → delete the source object (Yes/No) → Enter

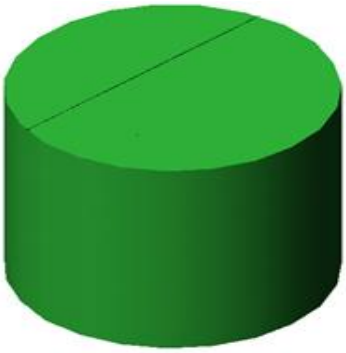


Figure 2

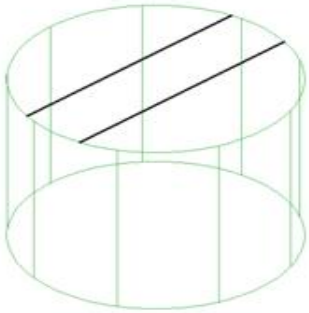


Figure 2

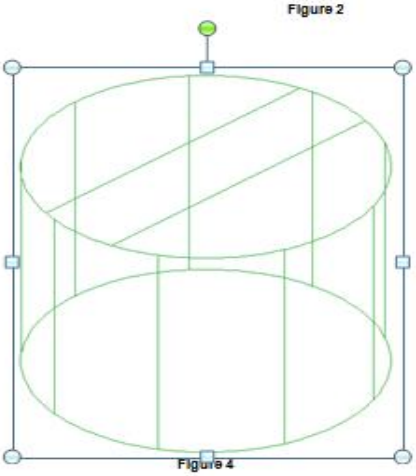
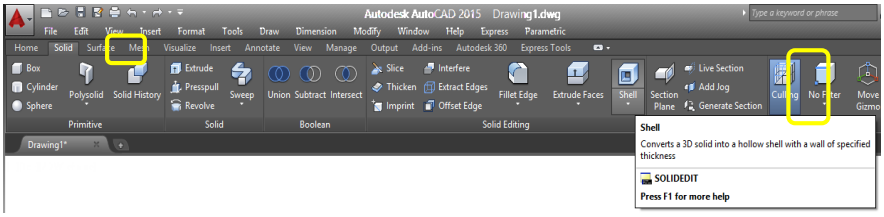


Figure 4

How to Apply “Shell” command on 3D solid object

Shell command is used to create a hollow, thin wall structure of a specified thickness. You can specify a constant wall thickness for all the faces. Also the desired faces can be removed as well.

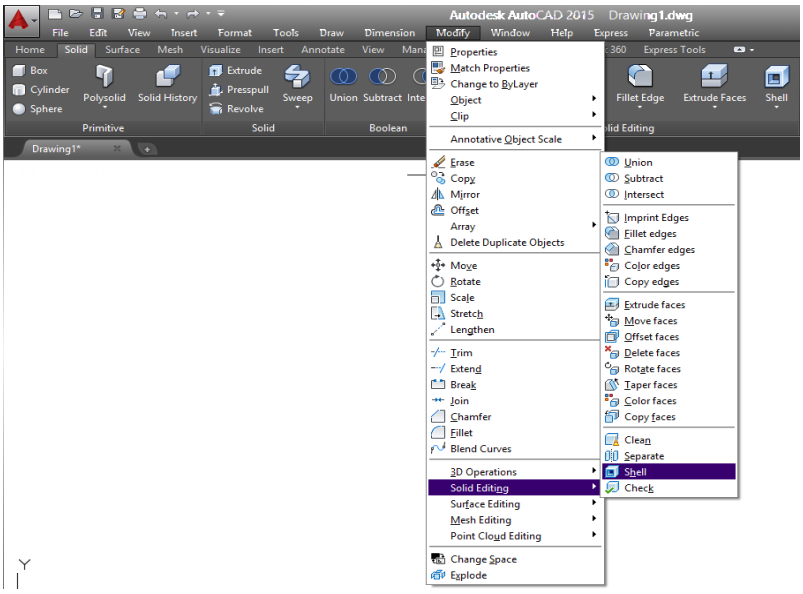
Shell command can be find out through both methods.



Through Ribbon

Through Main Manu

Menu Bar→Modify →Solid Editing →Shell



APPLICATION

Pick tool → Select 3D Solid →Select faces to remove → Enter→ put the wall thinness→ Enter

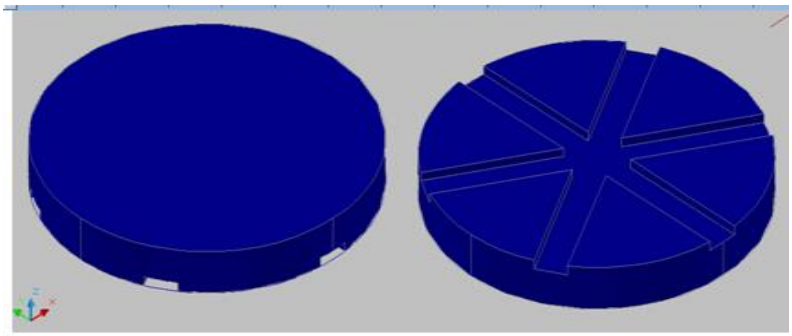


Figure 1(Top & Bottom)

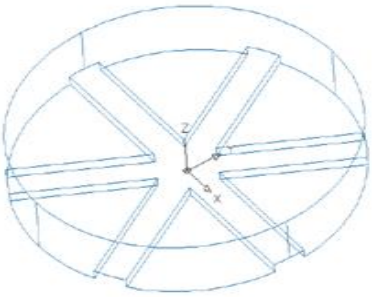


Figure 2

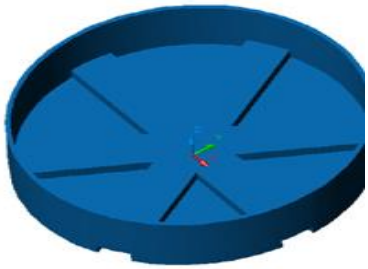


Figure 2

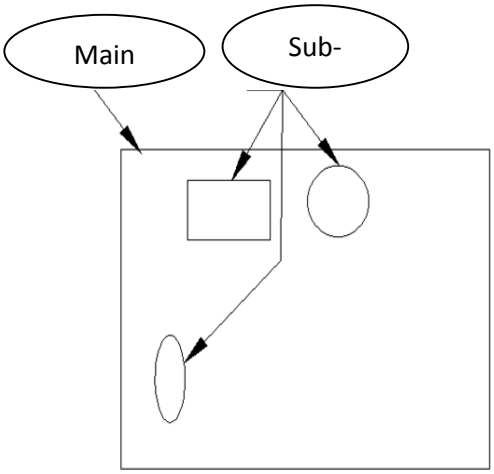
Learning Unit 2: Edit 3D Solid Objects

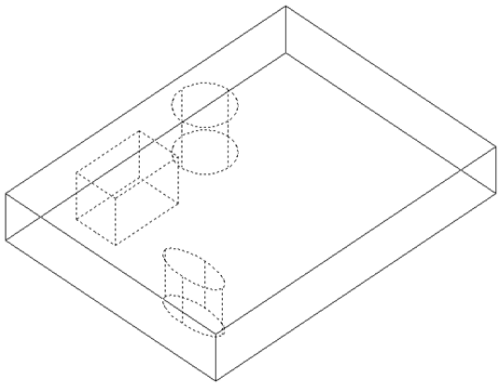
Overview

This unit provides information about editing 3D solid object, how to manipulate a selection set of more than one sub-object on any number of solids that include more than one type of sub-object. At the end of this unit you will be able to edit 3D solid objects by using different command e.g use of grip to change the size or shape, 3D move command will help you to move a set of object /sub-object along any axis, rotate command will be used for free or constrained rotation of object / sub-object along any axis up to specified angle and press pull command will be used to press or pull any bounded area on a object by coplanar lines and edges.

1. Manipulate Sub-object With "Ctrl" key

You can select one or more sub-objects in any number on 3D objects. The Selection set includes more than one type of sub-object. Press and hold Ctrl key to select sub-objects at the selection prompts of the MOVE, ROTATE, SCALE, and ERASE commands.





Do you know!

If you are unable to select the sub-object properly by holding Ctrl key then you can undo this step by pressing Ctrl + Shift keys.

APPLICATION

Open a drawing with 3D solid object and orient it in an Isometric view →Hold the Ctrl key and select the solid→Toggle to other sub-objects within the solid while Holding the Ctrl key→Click and drag to modify the solid.

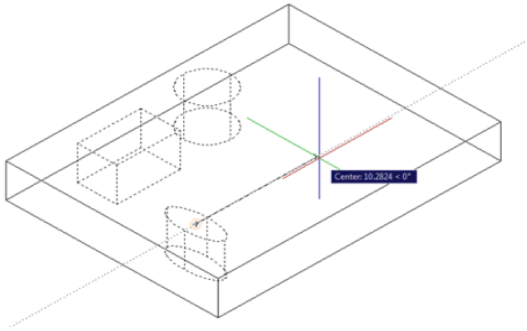


Figure -01

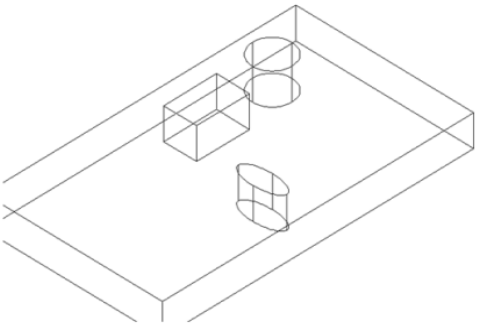


Figure -01

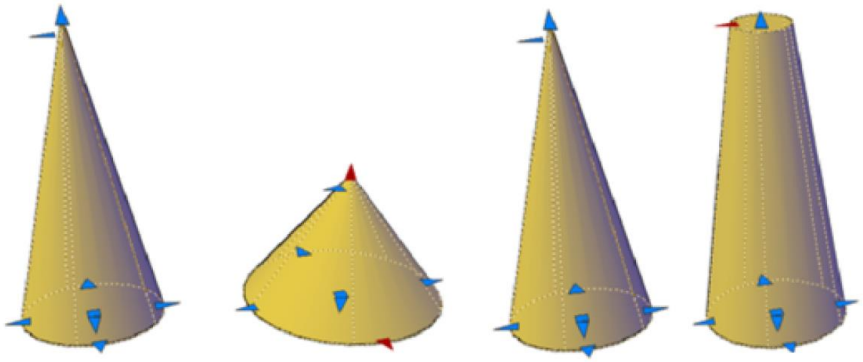
2. Use of Grips in 3D Primitive Solid

Grips is used to change the size and shape of some individual solids.

The method you use to manipulate the 3D solid depends on the type of object and the method used to create it.

Use of grip on Primitive Solid:- You can drag grips to change the shape and size of primitive solids.

For example, You can change the height and base radius of a cone without losing the overall cone shape. Drag the top radius grips to transform the cone to a flat-topped, cone.



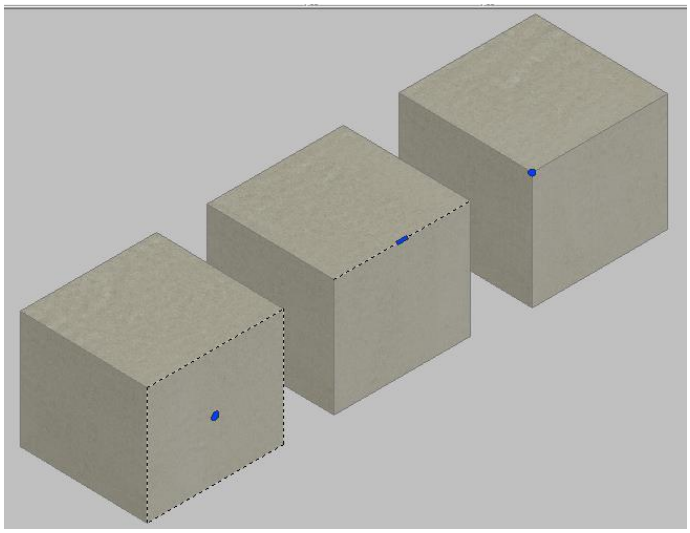
Select Sub-objects from Primitive Solid

Grip tools are icons that you use in a 3D view to constrain the movement or rotation of a selection set of objects along any axis or a plane. Press and hold Ctrl key to select a face, edge, or vertex of a 3D object. Selected sub-objects display different types of grips, depending on the sub-object type.

Grips are able to rotate selected sub-object in any direction.

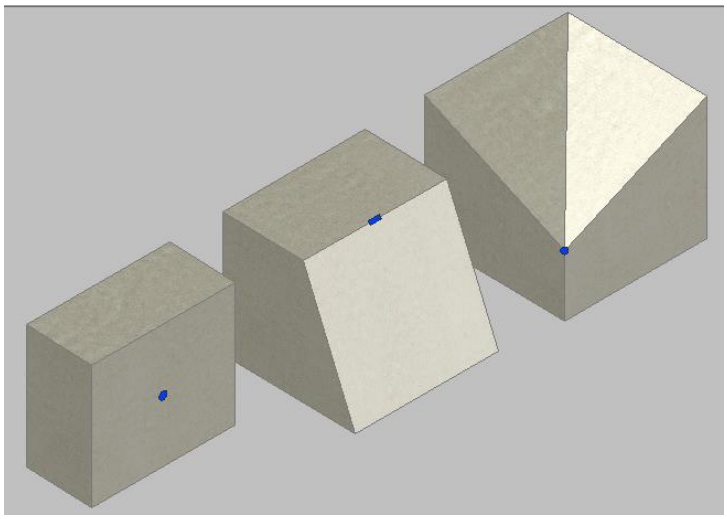
APPLICATION

Open a drawing with 3D solid object and orient in an isometric view→Hold the Ctrl key and select the sub-object on 3D solid Objects

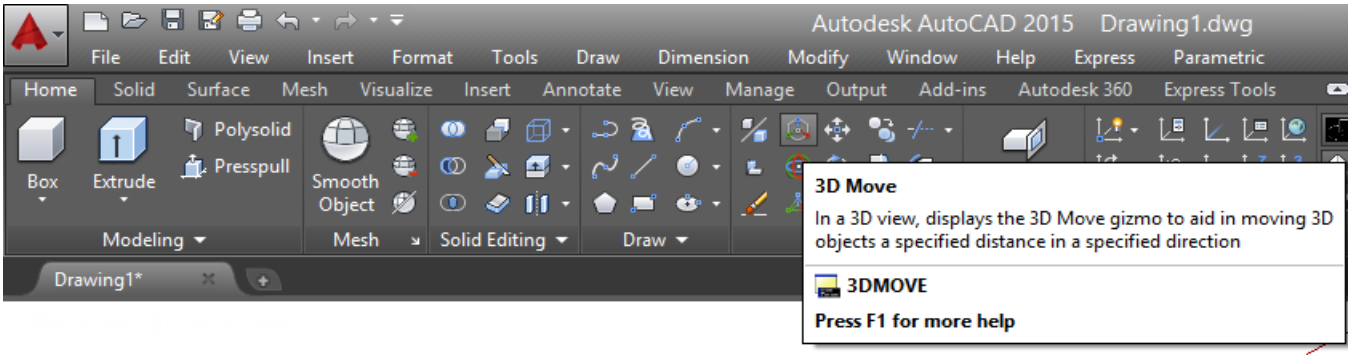


APPLICATION

Open a drawing with 3D solid object and orient in an isometric view→Hold the Ctrl key and select the solid→Toggle to other sub-objects within the solid while holding the Ctrl key→pick and drag to modify the solid



There are two types of grip tools:



1. **Move** (3DMOVE command)
2. **Rotate** (3DROTATE command

3. 3D MOVE

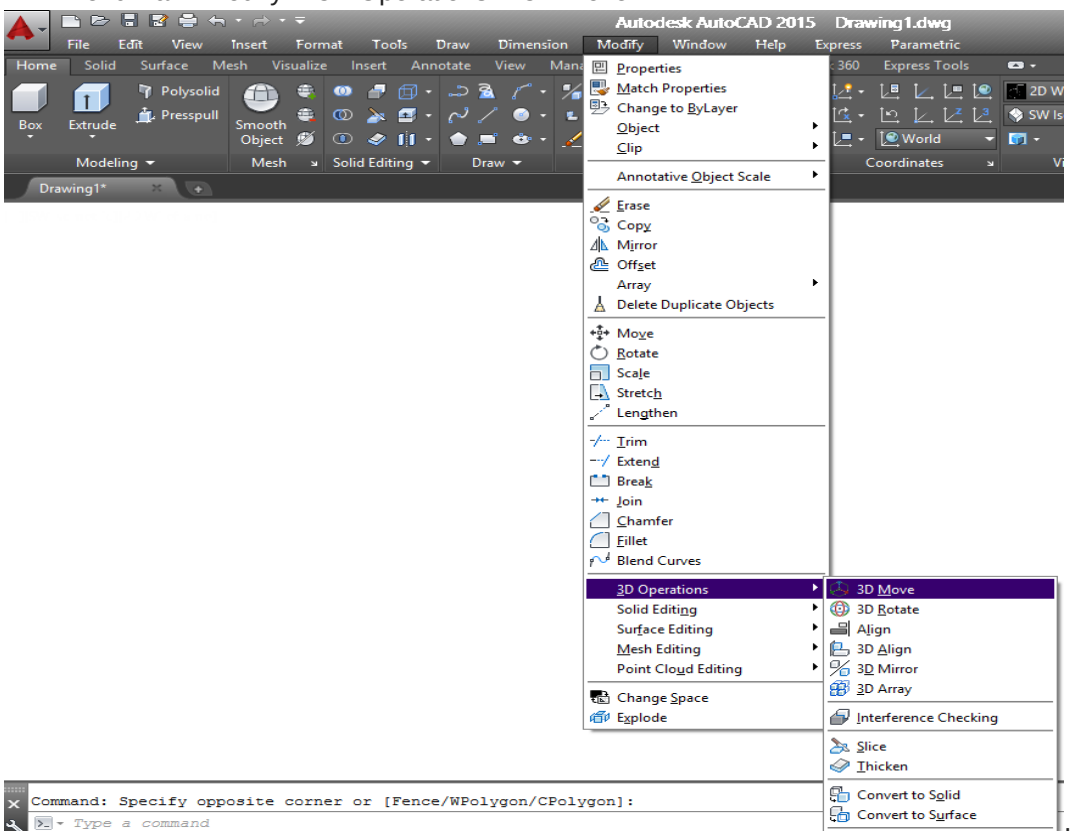
3D move command is used to move a selection set of objects or sub-objects to a specified distance in a specified direction or freely constrain the movement along any axis or plane.

3D Move command can be find out through both methods.

Through Ribbon

Through Main Manu

Menu Bar→Modify →3D Operations →3D Move

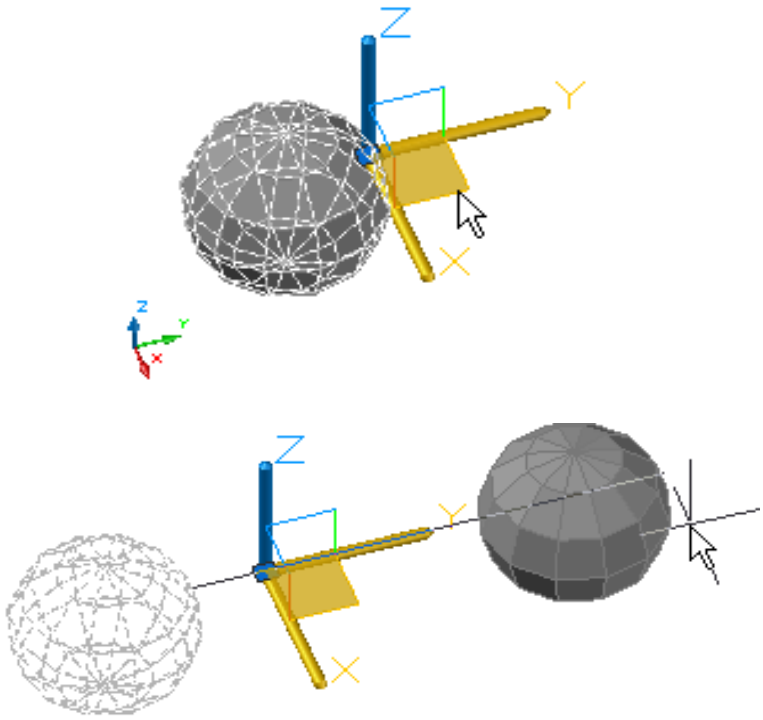


To move selected objects along a plane:

Move the cursor over the planar rectangle that intersects the plan and use to control the plane of constraint. When the rectangle turns yellow, click it.

APPLICATION

Pick Tool →Select the objects OR hold Ctrl key to select sub-objects (faces, edges, and vertices). →Release Ctrl key and press Enter. →The Move gizmo is displayed attached to the cursor→Click to place the Move gizmo, specifying the base point for the move. →Move the cursor over the plane →put the specify distance of the move.

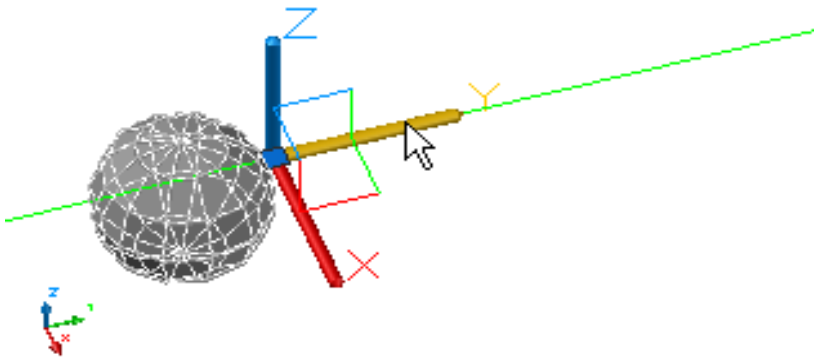


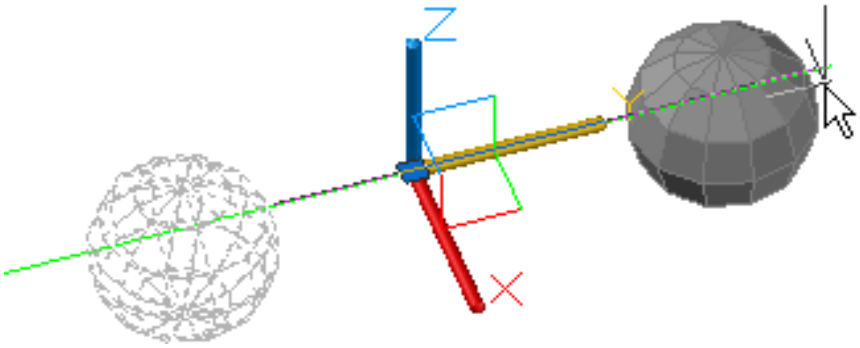
To move selected objects along an axis:

Move the cursor over an axis to control the gizmo until it turns yellow and the vector is displayed. Then click the axis handle

APPLICATION

Pick Tool →Select the objects OR hold Ctrl key to select sub-objects (faces, edges, and vertices). →Release Ctrl key and press Enter. →The Move gizmo is displayed attached to the cursor→Click to place the Move gizmo, specifying the base point for the move. → Handle on the gizmo until it turns yellow and the vector is displayed. Then click the axis handle.→Put the specified distance of the move.



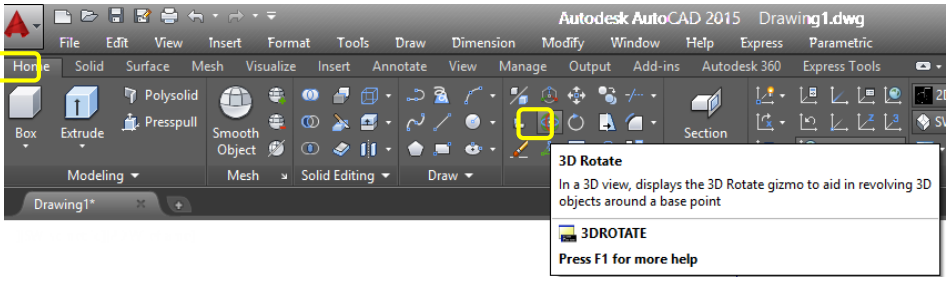


4. 3D Rotate

3D Rotate command helps to constrain or in rotation of 3D objects and sub-objects along any axis up to a specified angle

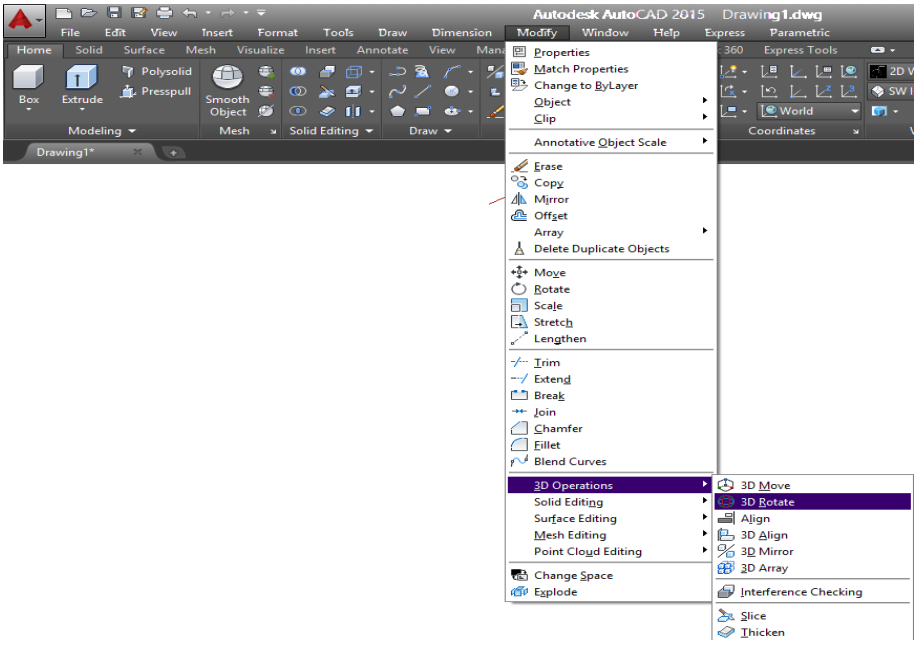
3D Rotate command can be find out through both methods.

Through Ribbon



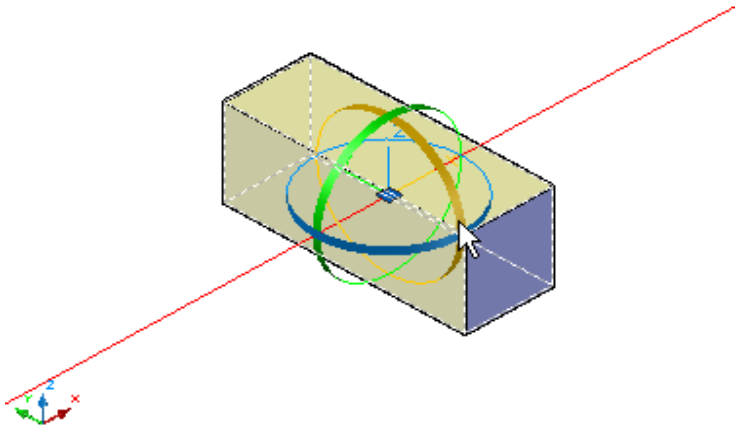
Through Menu Bar

Menu Bar→Modify →3D Operations →3D Rotate

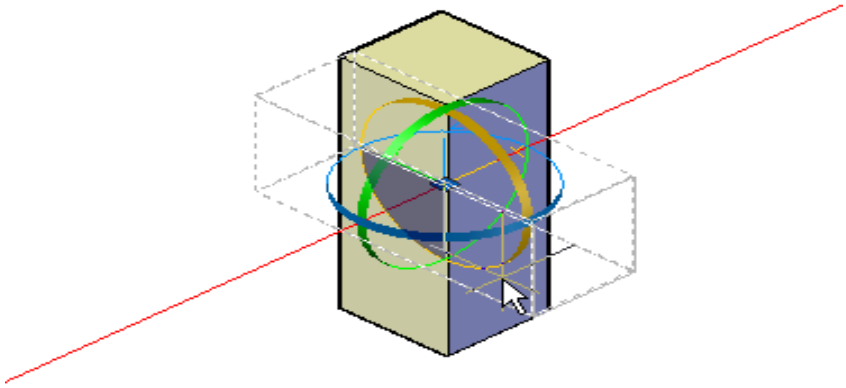


APPLICATION

Pick Tool →Select the objects OR hold Ctrl key to select sub-objects (faces, edges, and vertices) →Release Ctrl key and press Enter →The rotate gizmo is displayed attached to the cursor→ Click to place the rotating gizmo, specifying the base point for the move → However the cursor over an axis path on the gizmo until it turns yellow and the vector that represents the axis of rotation is displayed. → Specify angle of the rotation.



When you drag the cursor, the selected objects and sub-objects rotate about the base point along the specified axis. The gizmo displays the degree of rotation from the original position of the object as the object moves. You can click or enter a value to specify the angle of the rotation.



5. Press pull Command for 3D Solid & Bounded Areas

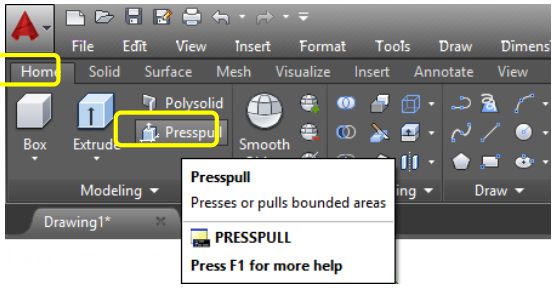
Press/Pull Command is used to press/pull any bounded area on an object by coplanar lines or edges.

Press pull command of 3D solids can be find out only through Ribbon.

Through Ribbon

Do you know!

For additional help about any tool press F1 Key.



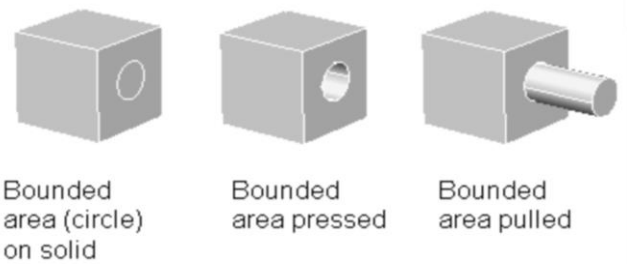
Press or pull a bounded area by holding Ctrl + Alt, or by clicking the Press pull button on the dashboard inside the area. The area must be bounded by coplanar lines or edges.

Then drag or enter a value to indicate the amount of extrusion. As you move the cursor, the extrusion changes dynamically.

You can also press Ctrl+Shift+E and click inside an area to initiate the press or pull activity

APPLICATION

Press and hold Ctrl + Alt→click any area bounded by coplanar lines or edges→drag the mouse to press or pull the bounded area→Click or enter a value to specify the height.

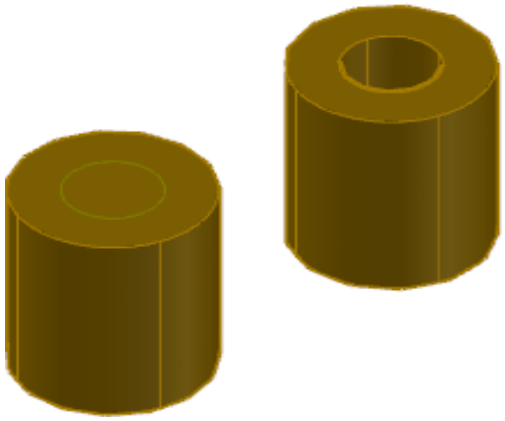


Work With Pressing or Pulling Bounded Areas or Objects

Push a Hole into 3D Solid:

APPLICATION

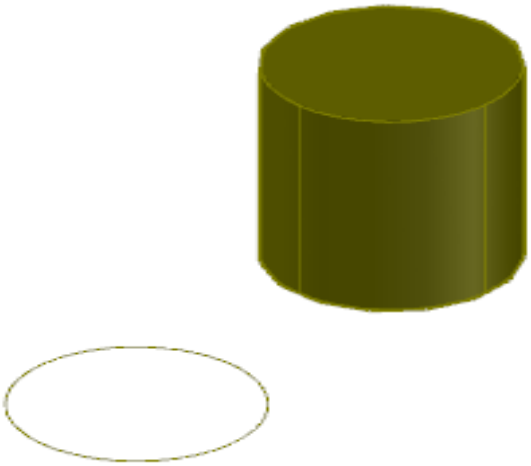
Press and hold CTRL + ALT keys→Click inside the closed shape or object and drag the cursor towards the interior of the 3D solid object to create the hole.



Create a Solid Object by Pulling a Closed Object

APPLICATION

Create the close object in polylines→Press and hold Ctrl + Alt keys→ Enter a value **or** click in the drawing area to indicate the height of the extrusion.



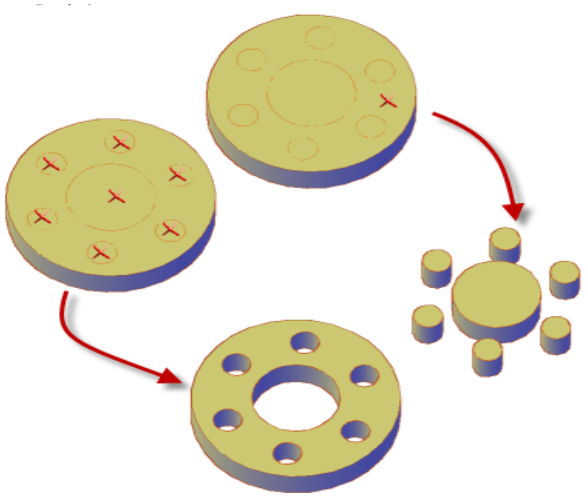
Remember

Objects used for multiple press or pull operation must support the same type of behavior (extrusion or offset).

Press or Pull Multiple Areas or Objects

APPLICATION

Create the close object in polylines on 3D Solid Faces → Press and hold Shift as you select the object, face, or area.→ Select the object, face, or area. Then enter **m** (Multiple) and continue to make selections. → Press Enter when the selection set is completed.



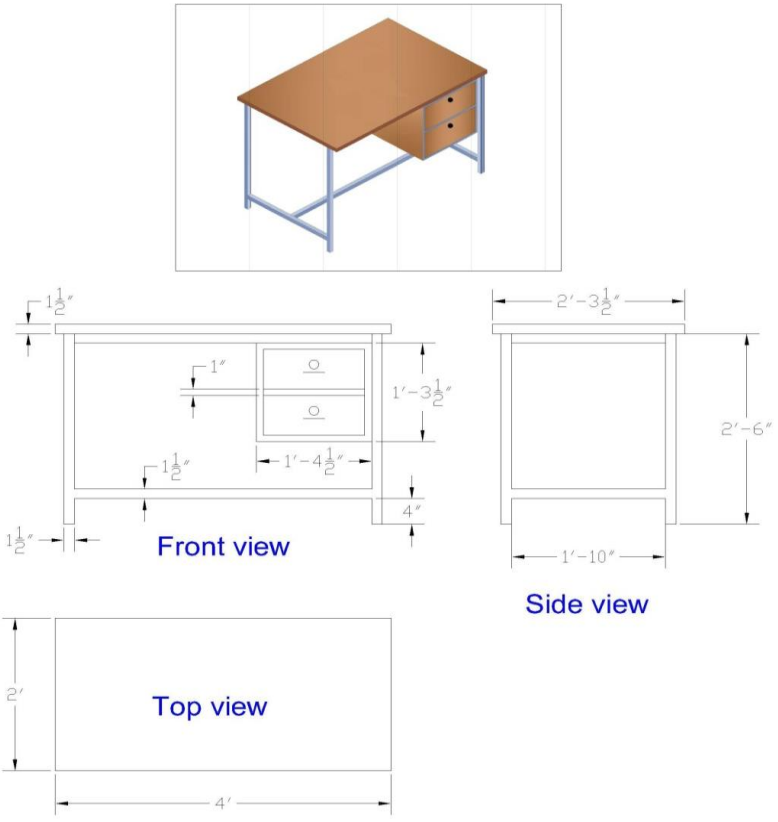
Learner's Activity

1. Draw a house plan according to following requirements.
(For Civil Department)

A. Plot Size 45'X60'(Regular)
B. Requirements

- **Two Beds with Attached Bath**
- **Kitchen**
- **Drawing Room**
- **Dinning Room**
- **T. V Loung**
- **Car porch**
- **Store**

2. Make Table in 3D according to specifications given below.
(For Mechanical Department)



Summary of Module

- In this module we have learnt about the modification and editing 3D solid faces. This module has described modifying solid faces using taper faces, extrude faces, delete faces, copy and color faces commands. We have learnt about imprint functionality on arc, circle, 2D & 3D polylines, ellipses splines regions bodies and 3D solid objects.

- We also have explored different methods to manipulate sub-objects in solids. Such as grips are used to change the size and shape of solids, 3D move command is used for free or constrain movement of sub-objects along any axis and plane. In the same way 3D rotate command describes about the rotation of sub-object along any axis up to a specified angle. We also have learnt how to press or pull a bounded area by coplanar lines or edges.

Frequently Asked Questions (FAQs)

FAQ 1:	What have we learned in this module?
Answer	To modify solid faces
FAQ 2:	Is there any difference between copy faces and extrude faces?
Answer	Yes, copy faces command is used to copy the face and edges of an object while extrude faces command is used to increase or decrease the length of any side of the object.
FAQ 3:	Which command is used to hollow the object?
Answer	Shell command
FAQ 4:	Which command is used to color the face of an object?
Answer	Color faces
FAQ 5:	What is the purpose of Ctrl key?
Answer	Witch key is used to manipulate the selected sub-object on the main object in 3D solid editing?
FAQ 6:	What is the location of modify solid tools?
Answer	Main menu→modify→solid editing→solid editing tools
FAQ 7:	How many options of extrude faces?
Answer	Two 1:height 2:path
FAQ 8:	Which key is used to activate presspull the command?
Answer	Ctrl + Alt keys
FAQ 9:	How can we pick the imprint command from menu?
Answer	Main menu→modify→solid editing→imprint
FAQ 10:	What is the location of 3D move and rotate command?
Answer	Main menu→modify→3D operation→3D move

Test Yourself!

Please mark the correct one from the given options. You can check your answer with the Answer Key at the end of this module.

1. What is the purpose of copy faces command?

a. Copy faces
b. Copy Edge
c. Copy object
d. Both a and b

2. From which menu, we can pick the 3D move?

a. Draw
b. Modify
c. Format
d. Dimension

3. What is the use of copy faces?

a. Faces extrude height
b. Copy faces
c. Move faces
d. None of these

4. How many options of extrude faces are available?

a. Two
b. Three
c. Five
d. Seven

5. What is the purpose of shell command?

a. To Cut the object
b. To change the plane
c. To hollow the object
d. None of these

Answer Key

MCQ Number	Correct Answer
1	d
2	b
3	b
4	a
5	c

AUTOCAD

Module-10

LEARNING GUIDE

National Vocational Certificate Level 3

Version 1 - July 2013

Module 10: Navigate Section and Merge Flat Objects from 3D Model

Learning Outcomes

After completion of this module, You will be able to:

- Navigate Section Objects
- Merge Flat Objects

Learning Unit 1: Navigate Section Objects

Overview

This unit describes the fundamentals of Navigate Section. After completion of this unit you will be able to understand to manipulate, generate 2D and 3D section and perform slice command on 3D objects.

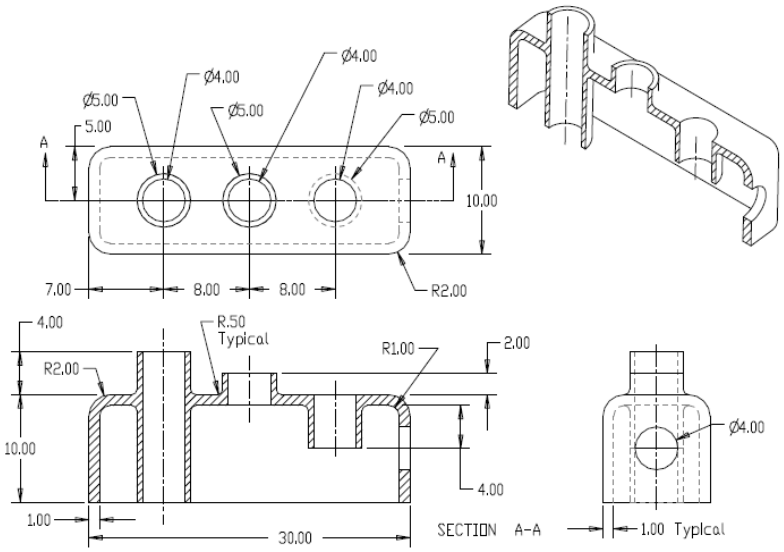
In this section of the lesson, you will learn to define section planes based on 3D models. You can use these section planes to view internal sections of 3D models, create 2D cross sections of models, create 3D cross-sectional models, and generate 2D geometry containing mass property information. You can use mass property information in strength and area calculations. Which are essential in determining items such as safety factors or material costs?

Point to be Noted

They are used if there is restricted space for section or dimensioning prevents the use of an ordinary rotated section.

1. What is The Section Plan of 3D Model?

Section objects have a transparent section plane indicator that acts as a cutting plane. Because it is transparent, it's a useful visual tool as it lets you to see the geometry on either side of the cutting plane. This plane can be easily placed and moved anywhere throughout a 3D model that is composed of 3D solids, surfaces, or regions (two-dimensional areas created from closed shapes or loops).

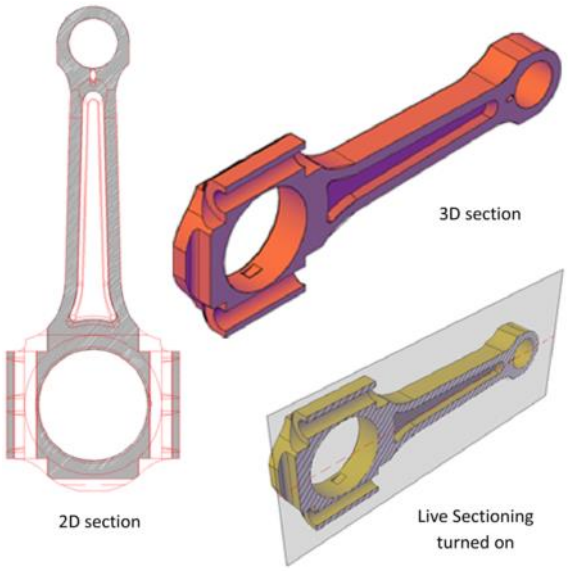


The section plane contains a section line where object properties for a section object are stored. You can have multiple section objects where each has different properties. For example, one section object can display a hatch pattern where the 3D model is intersected, while another section object can display a different line type for the boundary of the intersected area.

2D Sections

2D sections are created by cutting a number of objects with a section line and then creating a 2D section object from them. The section object is drawn without hidden and overlapping lines. You can edit a 2D section by changing its object display properties or its style display properties. The 2D section style lets you add to display

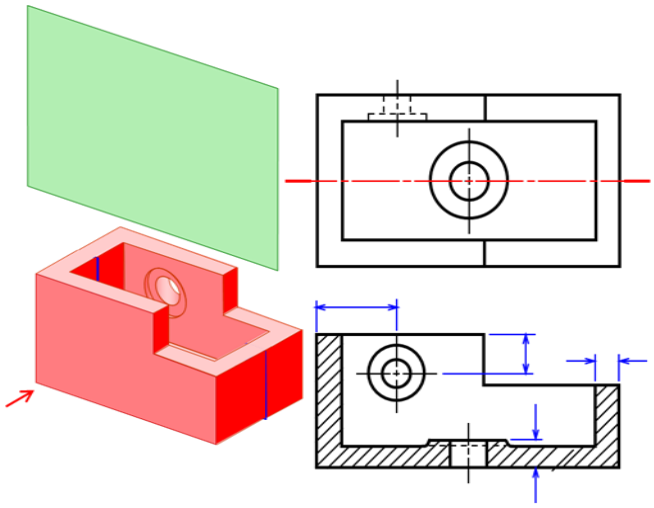
components to the display representation of the section and create rules that assign different parts of the section to different display components. You can control the visibility, layer, color, line type, line weight, and line type scale of each component. You can also choose whether materials, such as brick or concrete hatches, are used to display individual components of the section object or the 2D section style. Furthermore, you can use line work editing commands to assign individual lines in a 2D section to different display components, and merge geometry into a 2D section. You can dimension 2D sections.



3D Sections

3D sections are created by cutting a number of objects with a section line and then creating a 3D isometric section object from them. 3D sections do not use styles. However, you can control the display of subdivisions within 3D sections. Using the Hidden Line Projection command, you can create 2D hidden line projections of any view of the 3D section that you can explode and edit or hatch.

3D section in 3D view



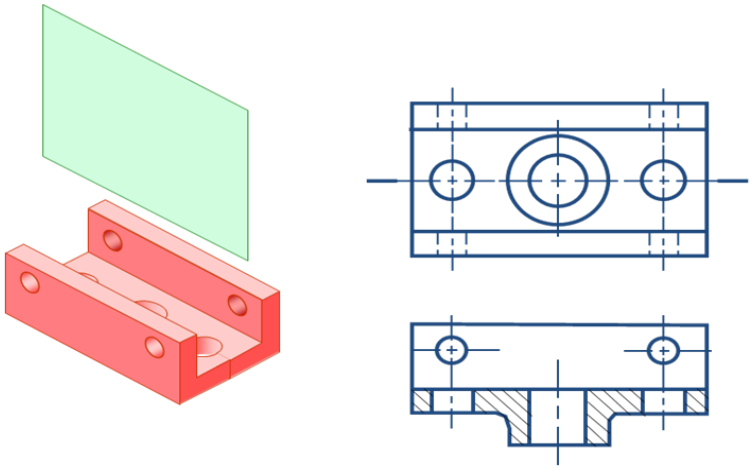
Types of sections

Choosing the type of section and location of the cutting plane.

1) **Full section**
2) **Offset section**
3) **Half section**
4) **Broken-out section**
5) **Removed section (detailed section)**
6) **Aligned Section**

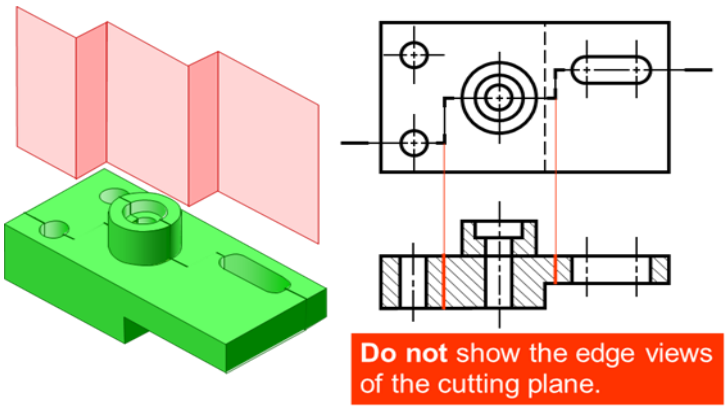
1) Full Section

This view is made by passing the straight cutting plane completely through the part dividing the object into two same parts.



2) Offset Section

This view is made by passing the zigzag cutting plane completely through the part.

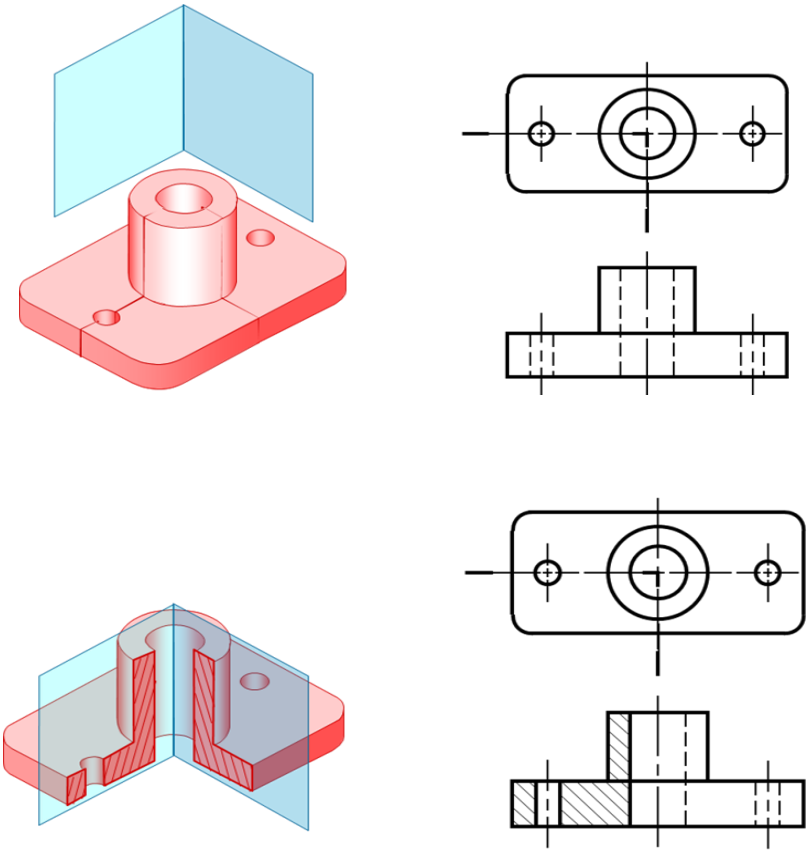


Remember

Do not show the edge views of the cutting .plane

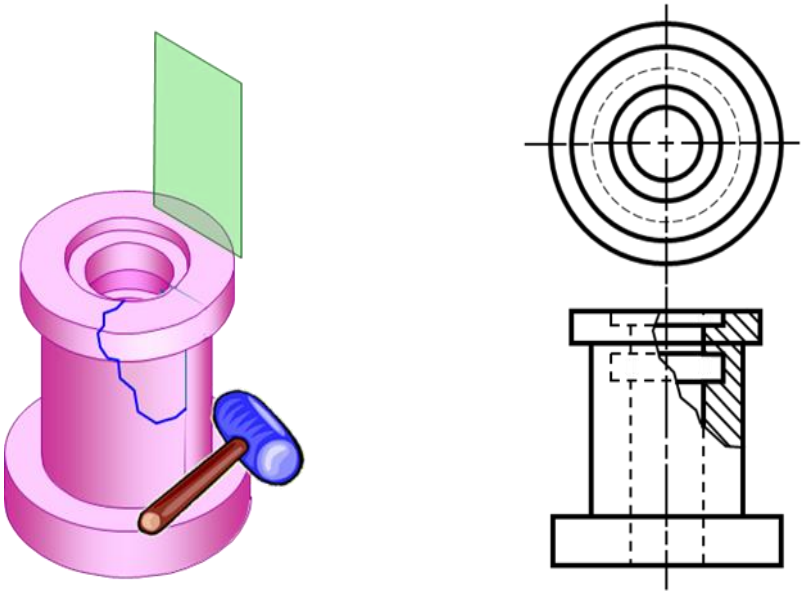
3) Half Section

The view is made by passing the cutting plane in such a way that it removes a quarter of it.



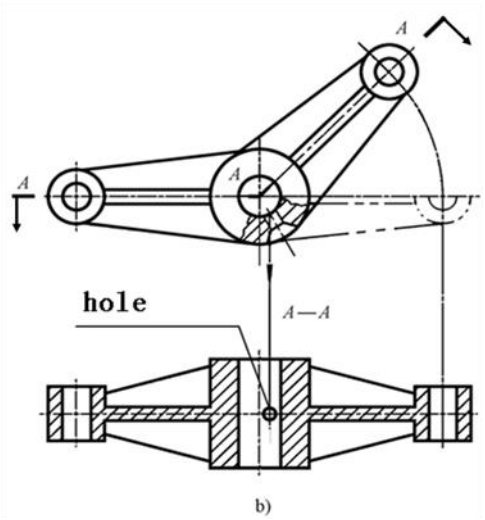
4) Broken-Out Section

This section is used when it is necessary to expose only a small portion of the internal shape of an object but not enough to make a full or half section.



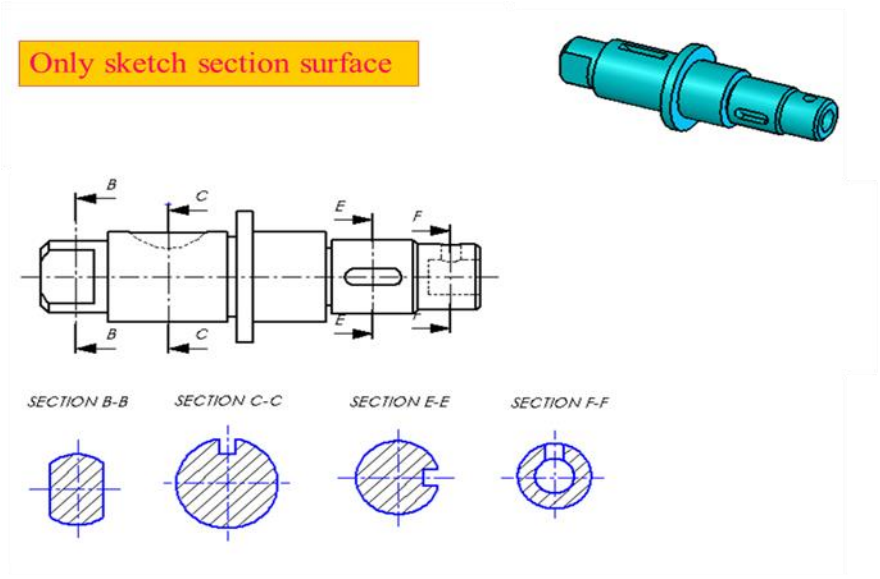
5) Aligned Section

Aligned section uses an angled cutting plane to pass through features which are at an angle. The plane and feature are then imagined to be revolved into the original plane and the section is projected from there.



6) Removed Section

This type of section is a revolved section drawn outside of the normal view. These sections are represented by a series of cutting planes and identifying letters (e.g. Section B-B, C-C, E-E, F-F and so on).



2. Sectioning Tools of 3D Objects

Location of Section Plane Tools for Sectioning of 3D solid Object. There are two ways for finding the location of sectioning tools.

1. Through Ribbon
2. Through Menu Bar

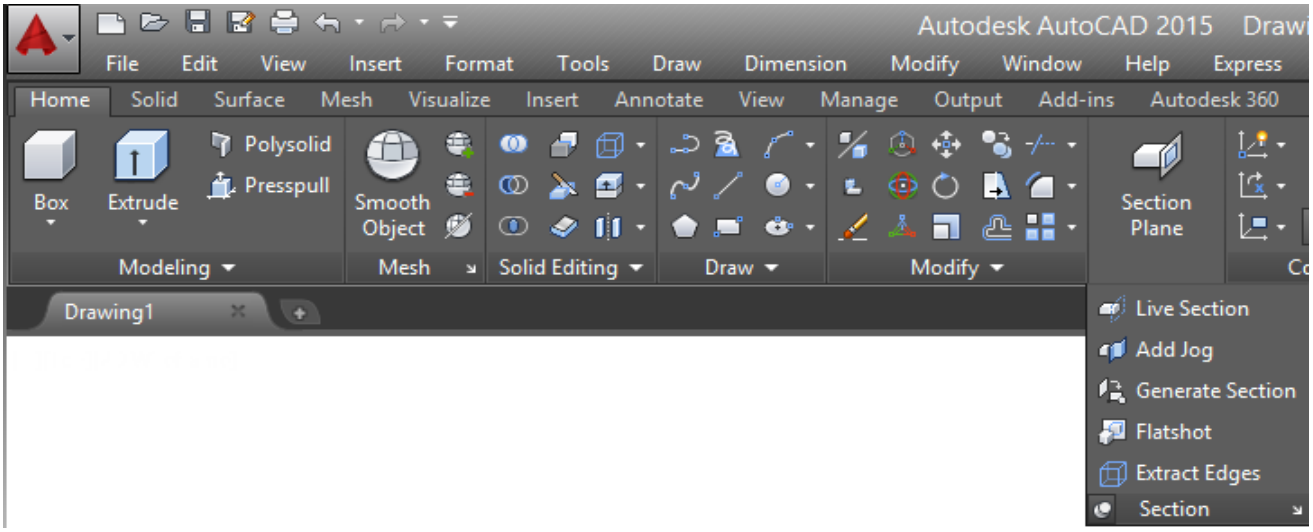
Section Plane Tools through Ribbon

Remember

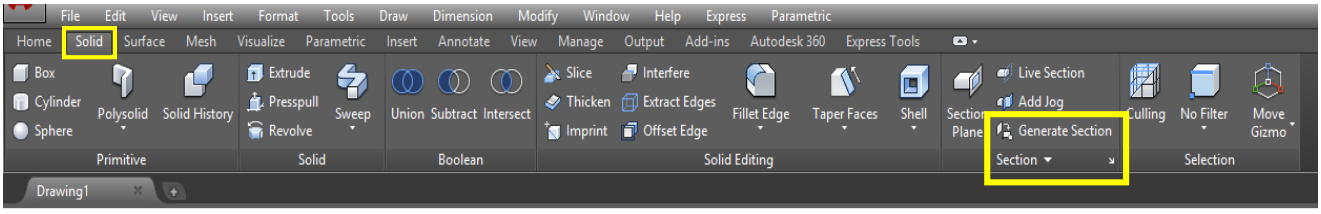
These tools are also available in the Section panel on the Solid ribbon tab.

Through Ribbon Section Plane Tools

You must create a section plane, which you can do by using the Section Plane tool in the Section panel of the Home ribbon tab (when working in the 3D Modeling workspace).



Through Solid Tab



Tools in the Section Tab

Following tools are available in the section tab.

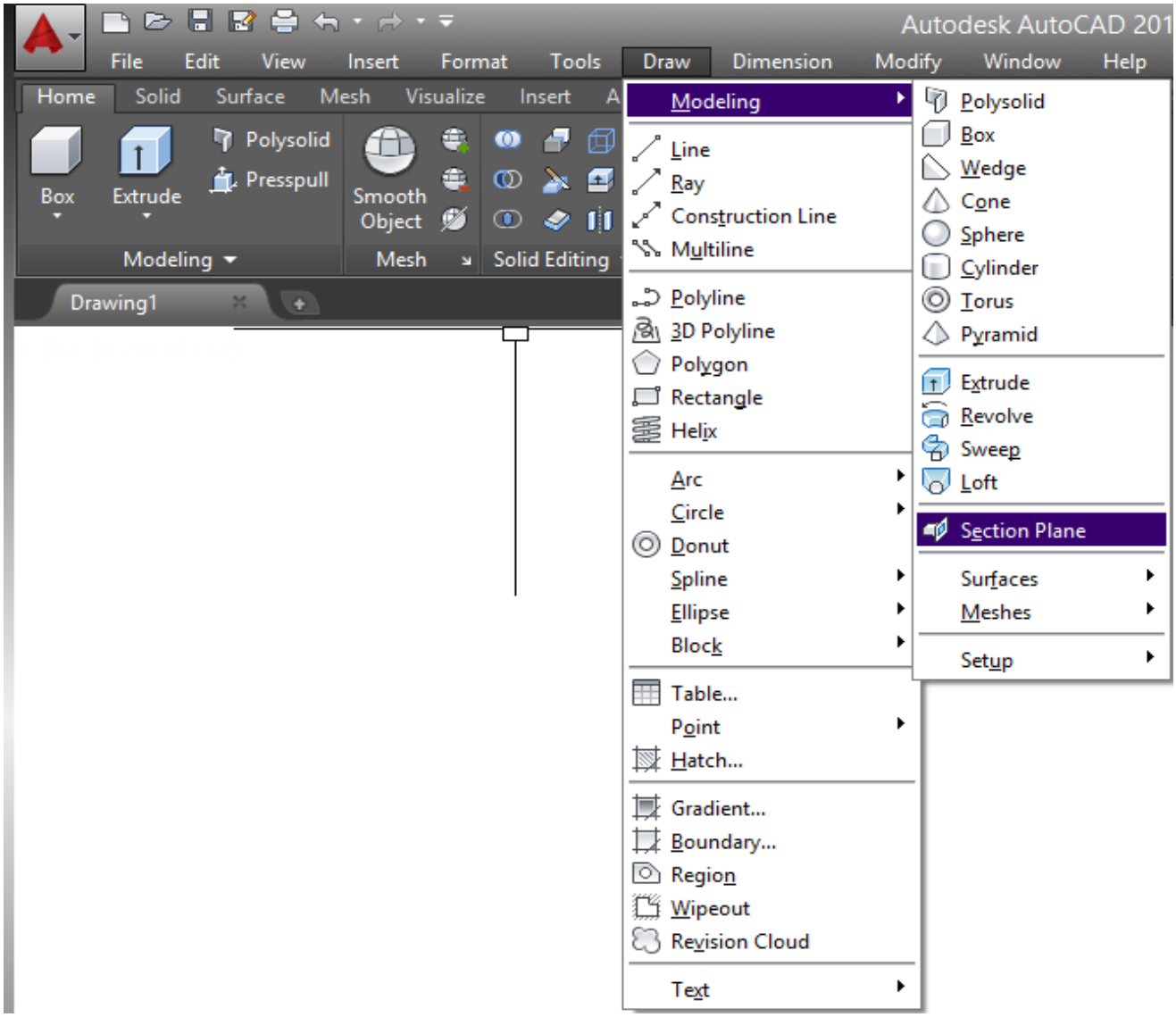
Live Section	Turns on live sectioning for a selected section object (**LIVESECTION**).
Add Jog	Adds a jogged segment to a section object (**SECTIONPLANEJOG**).
Generate Section	Saves selected section planes as 2D or 3D blocks (**SECTIONPLANETOBLOCK**).
Section Plane Settings	Saves selected section planes as 2D or 3D blocks (**SECTIONPLANESETTINGS**).

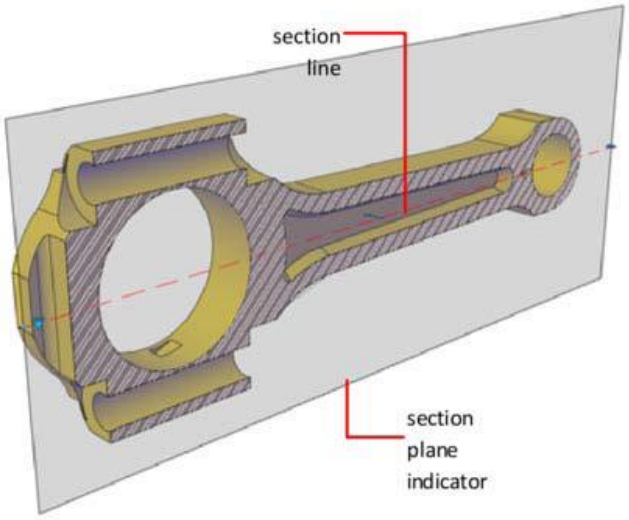
3. Use of Section Plane Command

Section plane objects create sections of 3D solids, surfaces, meshes, and point clouds. Use live sectioning with section plane objects to analyze a model, save sections as blocks for use in layouts, and extract 2D geometry from point clouds.

Through Menu Bar Section Plane

Menu Bar →Draw →Modeling→Section Plan


Autodesk AutoCAD 201
File
Edit
View
Insert
Format
Tools
Draw
Dimension
Modify
Window
Help
Home
Solid
Surface
Mesh
Visualize
Insert
Box
Extrude
Polysolid
Presspull
Smooth Object
Modeling
Mesh
Solid Editing
Drawing1
Modeling
Line
Ray
Construction Line
Multiline
Polyline
3D Polyline
Polygon
Rectangle
Helix
Arc
Circle
Donut
Spline
Ellipse
Block
Table...
Point
Hatch...
Gradient...
Boundary...
Region
Wipeout
Revision Cloud
Text
Polysolid
Box
Wedge
Cone
Sphere
Cylinder
Torus
Pyramid
Extrude
Revolve
Sweep
Loft
Section Plane
Surfaces
Meshes
Setup


section line
section plane indicator

APPLICATION

Pick Tool →Draw the Section Line on Object → Double Click on Section Plan Indicator→Show the Section of Object

Section Planes States

When you create a section plane, the state it is in, by default, depends on how the plane was originally created.

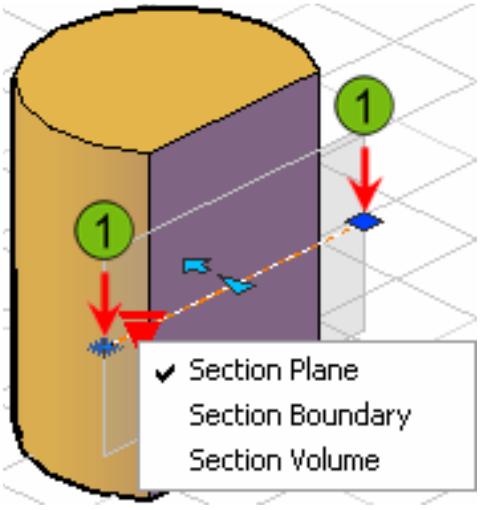
You can modify the section plane state at any time by selecting a different state from the grip menu.

There are three different section plane states:

i. Section Plane
ii. Section Boundary
iii. Section Volume

i. Section Plane:

The section plane extends infinitely.



Remember

1

Plane grips Drag to adjust the length of the section plane.

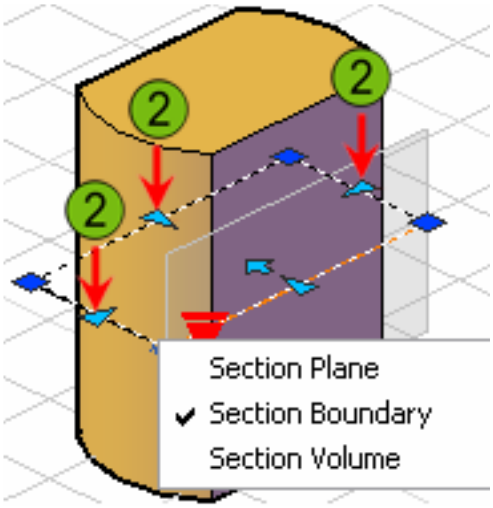
ii. Section Boundary:

The section plane object contains side planes and a back plane extending infinitely, and can section a model from all four sides.

Remember

2

Boundary grips:
Drag to adjust the effective boundary area of the section.



iii. Section Volume:

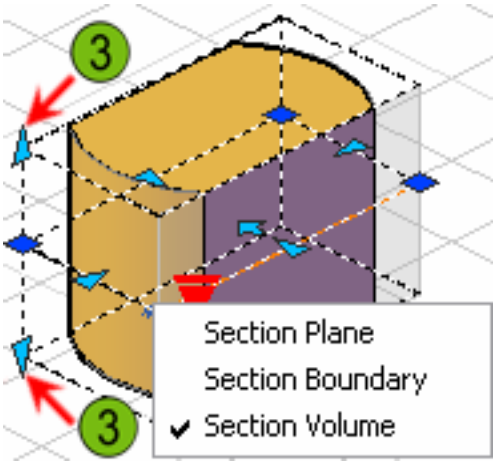
The section plane contains side planes, a back plane, and top and bottom planes. The model can be sectioned on six sides. The following images illustrate the effect of changing the section plane state.

Remember

3

Volume grips

Drag to adjust the effective volume of the section



Options of Section plane

1. **Face or Any Point to Locate Section Line**
2. **Draw section**
3. **Orthographic**

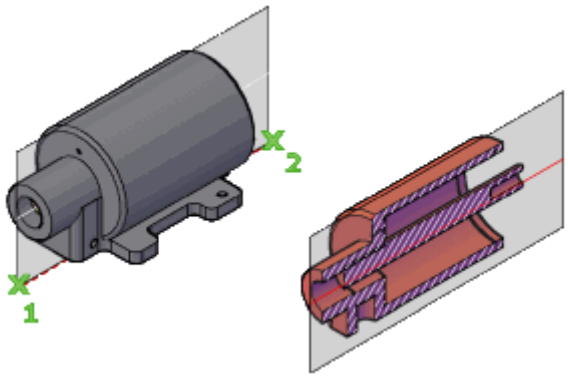
1. Face or Any Point to Locate Section Line

Specifies a face to establish the plane of the section object. Alternatively, you can select any point on the screen that is not on a face to create a section object independent of the solid or surface. The first point establishes a point around which the section object rotates.

- Through point. Sets a second point that defines the plane of the section object.

APPLICATION

Pick Tool →Draw the Section Line on Object → Double Click on Section Plan Indicator→Show the Section of Object

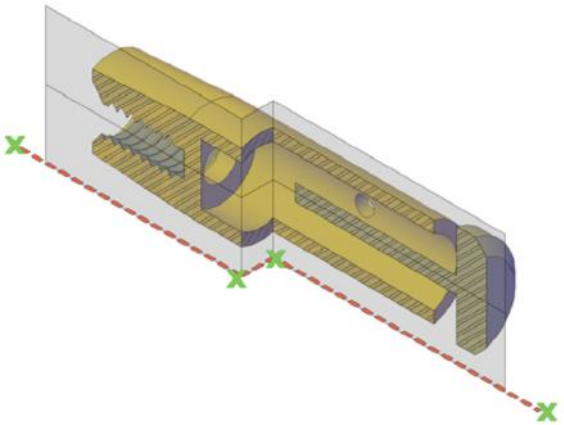


2. Draw section

Define the section object with multiple points to create a section line with jogs.

APPLICATION

Pick Tool →D enter (For Draw the Section Plan) → Draw the Section Plan→Double Click on Section Plan Indicator→Show the Section of Object



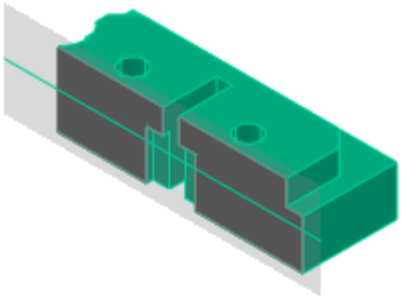
3. Orthographic

A section object that contains all 3D objects is created with the specified orientation relative to the UCS (not the current view). This option creates a section object in the Section Plane state with live sectioning turned on.

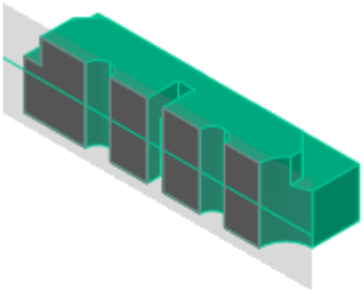
Align section to set the position of the section object to align with one of the following UCS orientations that you specify:

- Front
- Back

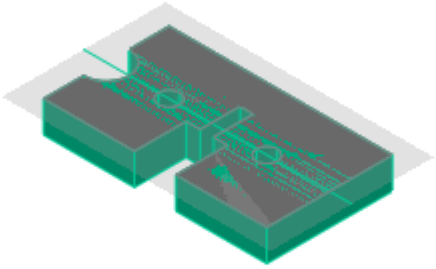
- Right
- Left
- Top
- Bottom



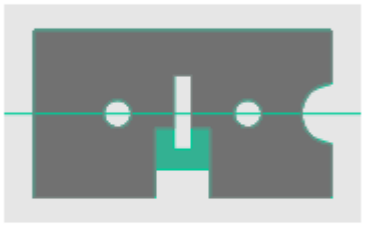
Front



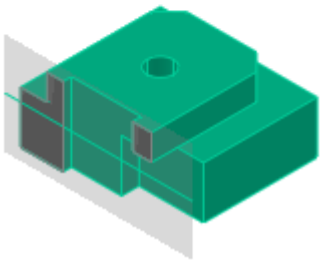
Back



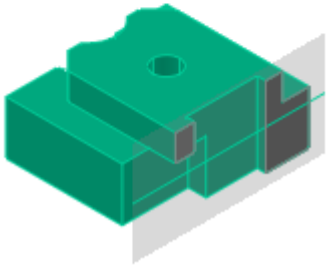
Top



Bottom



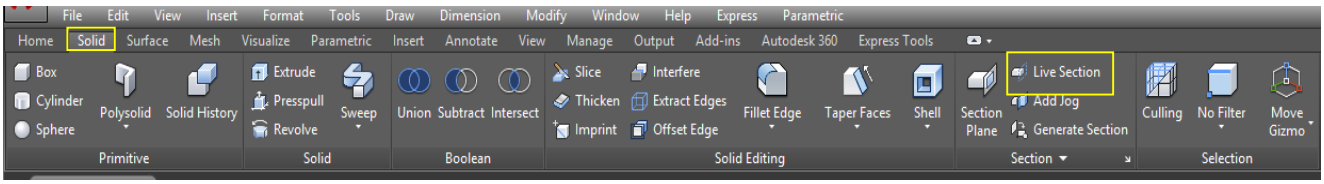
Left



Right

4. Live Section

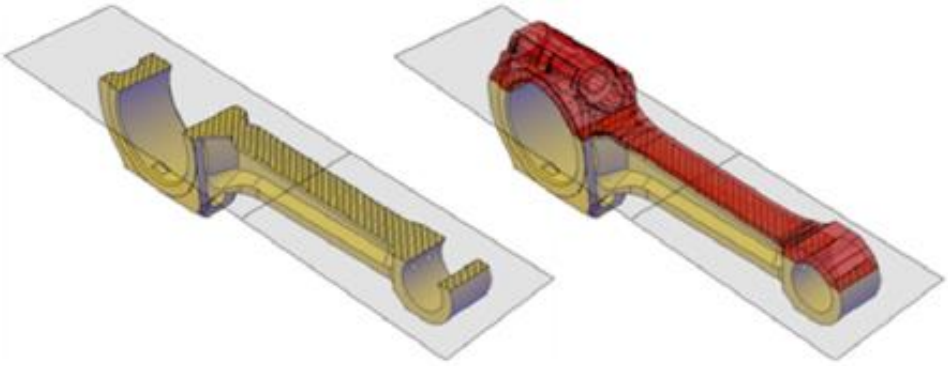
Even after the section plane has been positioned, it isn't actually cutting anything yet. To have the section plane show the cross section, you need to turn on live sectioning. You can do this one of two ways. If you click the Live Section tool, available by expanding the Section panel of the Home ribbon tab.



You select the section plane to display a cross section through the model. You can also select the section plane, right-click, and choose Activate Live Sectioning from the shortcut menu. When turned on, the section plane cuts through the model, displaying it as a cross section. When turned off, the section plane remains visible but the model is no longer shown in cross section.

APPLICATION

Pick Tool →Select Section Plan Indicator

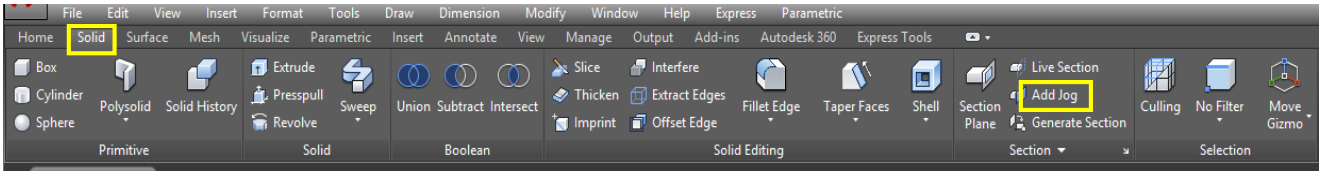


cut-away geometry not displayed

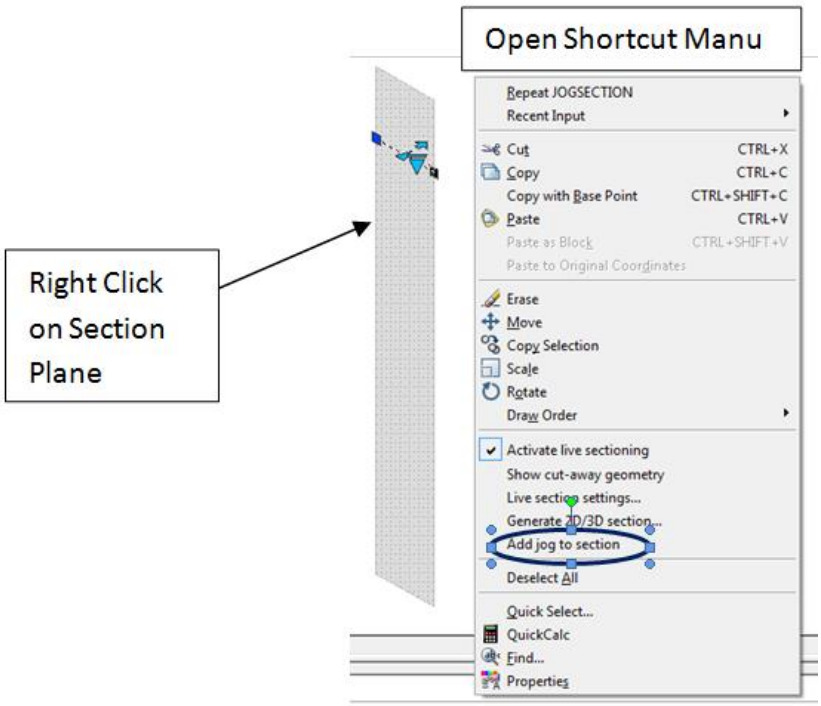
cut-away geometry displayed

5. Adding Jogs to a Section Plane

When you first create a section plane, it is a single plane that follows the path of the section line. But sometimes the 3D models you create are not symmetrical. Sometimes you may want to jog the section so that it cuts through the most interesting portions of the 3D model. This is quite easy to do using the Add Jog tool.



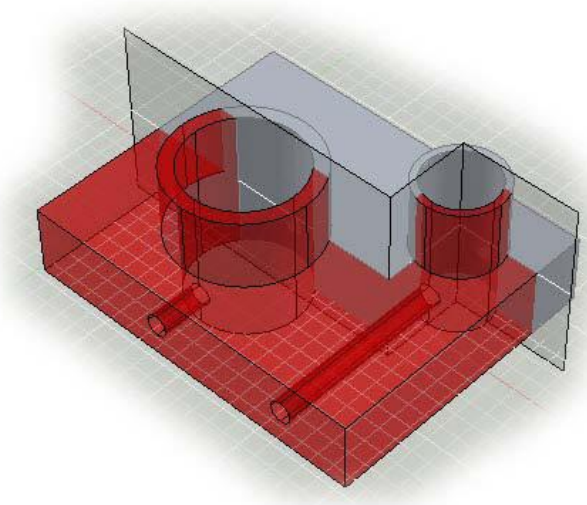
You can add a jog to a section plane by using the Add Jog tool (SECTIONPLANEJOG command), available by expanding the Section panel, or by right-clicking the section plane and choosing Add Jog to Section from the shortcut menu.



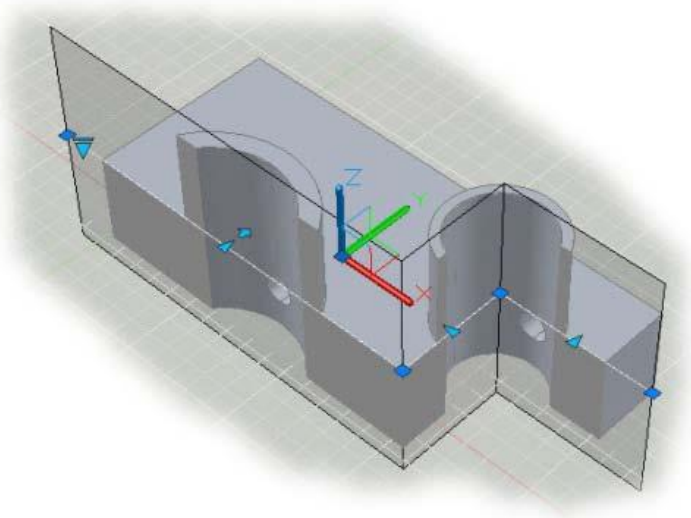
After starting the command and selecting the section plane now you have to specify a point on the section line to add the jog. It is often helpful to change the orientation of the view so that you can better see where you are positioning the jog. Click on the section plane where you to add the jog.

APPLICATION

Right Click on Section Plane →Open Shortcut Menu→Pick the Jog Tool →Select point on section line to add jog.

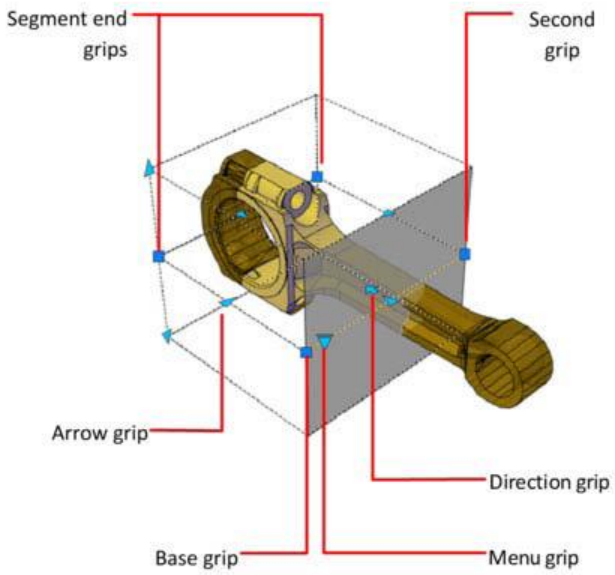


The Nearest object snap is temporarily turned on to help you place the jog. The jog is immediately created perpendicular to the selected section.



6. Manipulate the Section using Grips

Changing the section states by using grips to modify section objects. When you select a section plane of object to visible the Section plan grips, including a Menu grip and the Direction grip. If you click the Menu grip, you can change the display of the section object to one of three possible display states:



You can use these grips to move and resize the section object to adjust the location, length, width, and height of the cutting area.

- **Base grip:-**

  Acts as the base point for moving, scaling, and rotating the section object. It always displays adjacent to the Menu grip.

- **Second grip:-**

Rotates the section object around the base grip.

- **Menu grip:-**

  Displays a menu of section object states, which control the display of visual information about the cutting planes.

- **Direction grip:-**

  It controls the viewing direction of the 2D section click the direction grip in order to reverse the viewing direction of the section plane

- **Arrow grip:-**

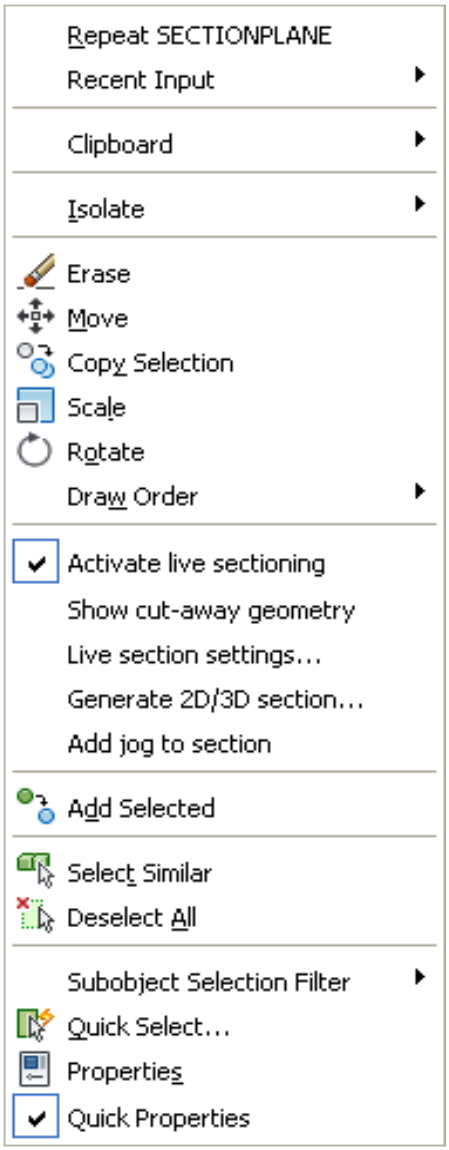
  It modifies the shape and position of the section plane. Only orthogonal movements in the direction of the arrow are permitted.

- **Segment end grips:-**

  You cannot move segment end grips so that segments intersect. Segment end grips are displayed at the endpoints of jogged segments.

7. Section Shortcut Menu Command

Section Plane Shortcut Menu



Remember

All these command have been discuss in detail in pervious module

Section Plane Shortcut Menu command

1. Erase
2. Move
3. Copy
4. Scale
5. Rotate
6. Draw Order

- Bring to front
- Send to back
- Bring Above Object
- Send Under Object

1) Erase Command

It is used to erase the Section Plane.

2) Move Command

This command is used to move the Section Plane on Object.

3) Copy Command

This command is used to copy the section plane.

4) Scale Command

It is used to increase & discuses the size of section plane.

5) Rotate Command

It is used to rotate the section plane with angle.

6) Draw Order

a) Bring to front

It is used to display of section plane in front of object.

b) Send to Back

It is used to hide the section plane behind the object.

c) Bring above Object

It is used to display the object in front of section plane.

d) Send under object

It is used to hide the object behind the section plane.

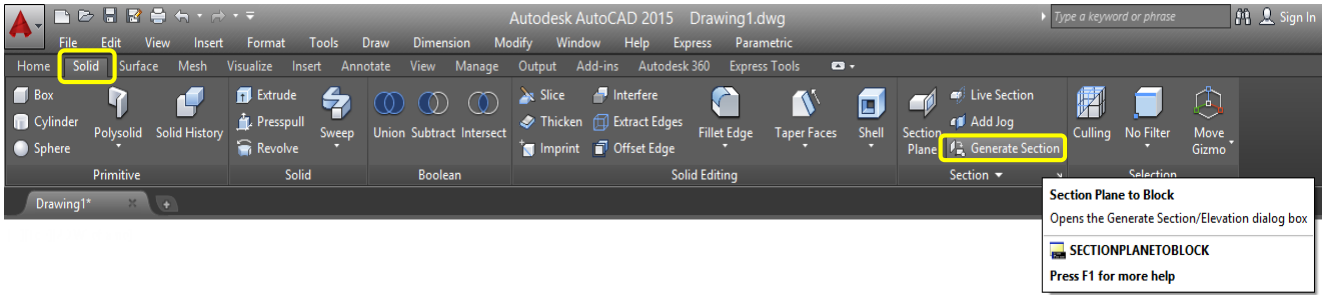
8. Generate 2D & 3D Section

Generates 2D & 3D section tools for 3D solid object. There are two ways for finding the location of sectioning tools.

1. **Through Ribbon**
2. **Through Shortcut Menu**

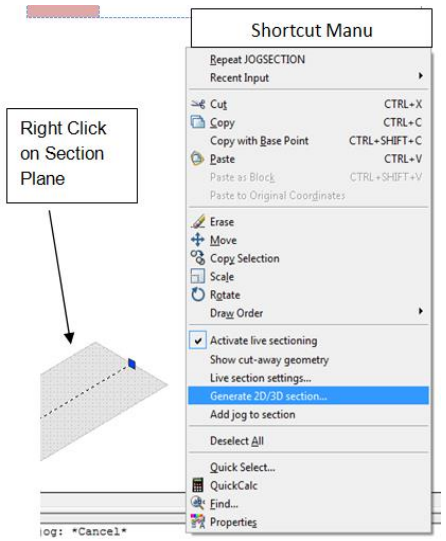
Through Ribbon

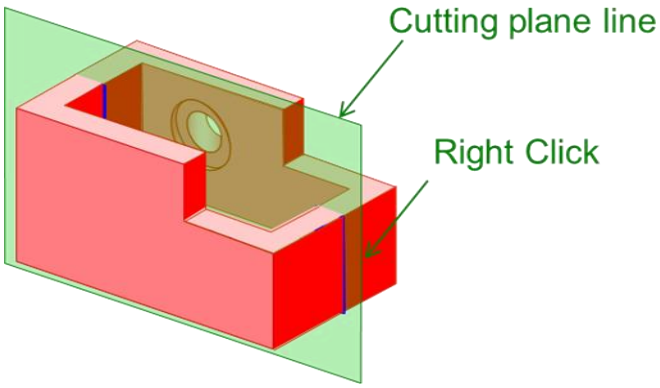
Generate Section, appears in the section panel of the solid tab.



Once you have created your section plane, you can generate 2D sections or elevations and 3D sections of your model. This option is on the shortcut menu when you select an existing section.

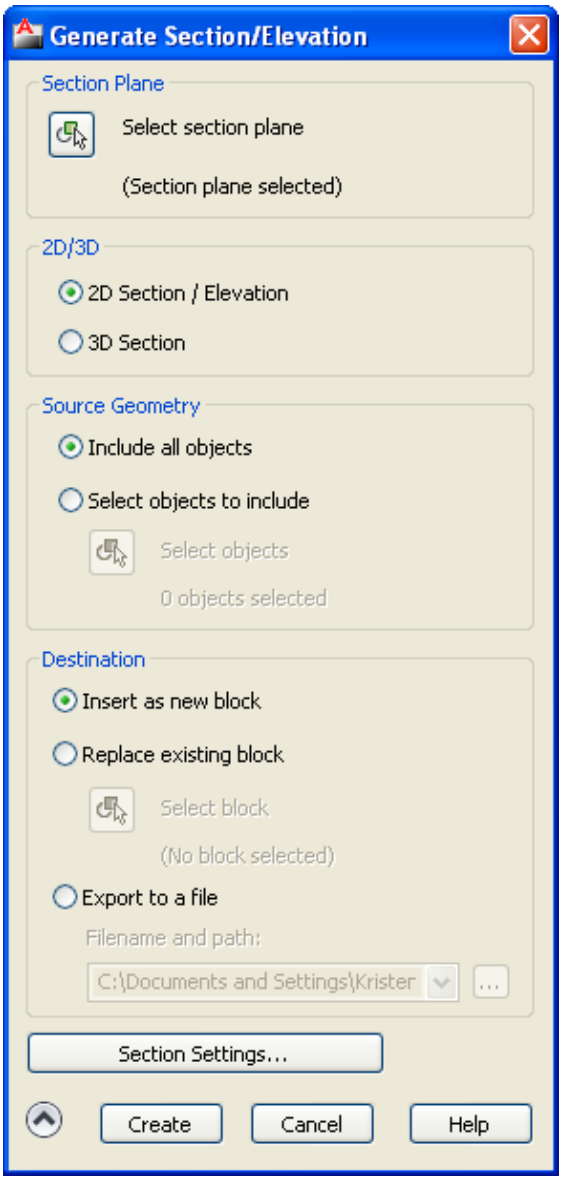
Through Shortcut Menu





You can also select a section object, right-click, and then choose Generate 2D/3D Section from the shortcut menu.

When you use this tool, AutoCAD displays a Generate Section/Elevation dialog.



List of Options:

1) Section Plane

- Select Section Plane

  Close the dialog box so that you can select a section plane to modify or save as a block.

2) 2D/3D

- 2D Section/Elevation

  Creates and saves or inserts a block that represents a 2D cross section.

- 3D Section

  Creates and saves or inserts a cutaway 3D block.

3) Source Geometry

- Include All Objects

  Includes all 3D objects (3D solids, surfaces, and regions) in the drawing, including objects in xrefs and blocks.

- Select Objects to Include

  Specifies that you will manually select the 3D objects (3D solids, surfaces, meshes, and regions) from which to generate a section.

- Select Objects

  Closes the dialog box temporarily while you select the objects for the section. When you finish selecting objects, press Enter to redisplay the Generate Section/Elevation dialog box.

- Objects Selected

  Indicates the number of objects selected.

4) Destination

- Insert as New Block

  Inserts the generated section as a block in the current drawing.

- Replace Existing Block

  Replaces an existing block in the drawing with the newly generated section.

- Select Block

  Close the dialog box temporarily while you select the block to be replaced in the drawing. When you finish selecting the block, press Enter to redisplay the Generate Section/Elevation dialog box.

- Export to a File

  Save the section to an external file.

- Filename and Path

  Specifies a file name and path where the section will be saved.

5) Section Settings

- Opens the Section Settings Dialog Box.

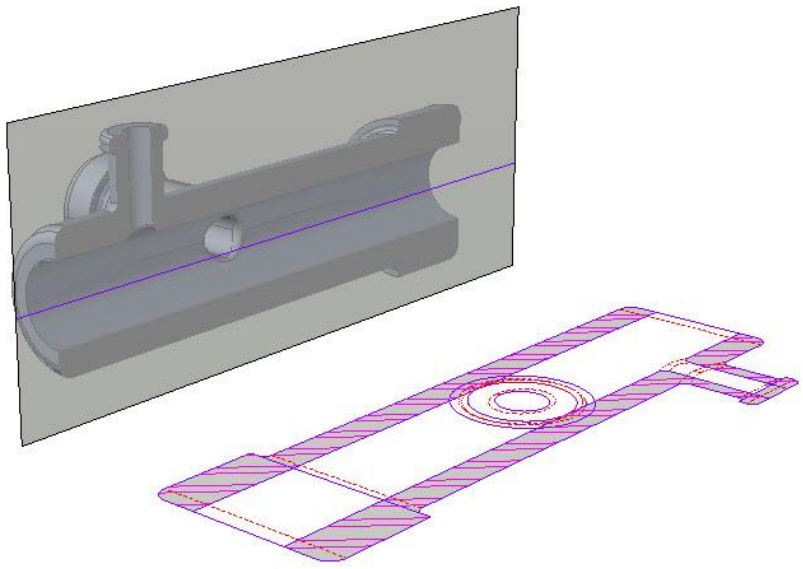
6) Create

- Creates the section

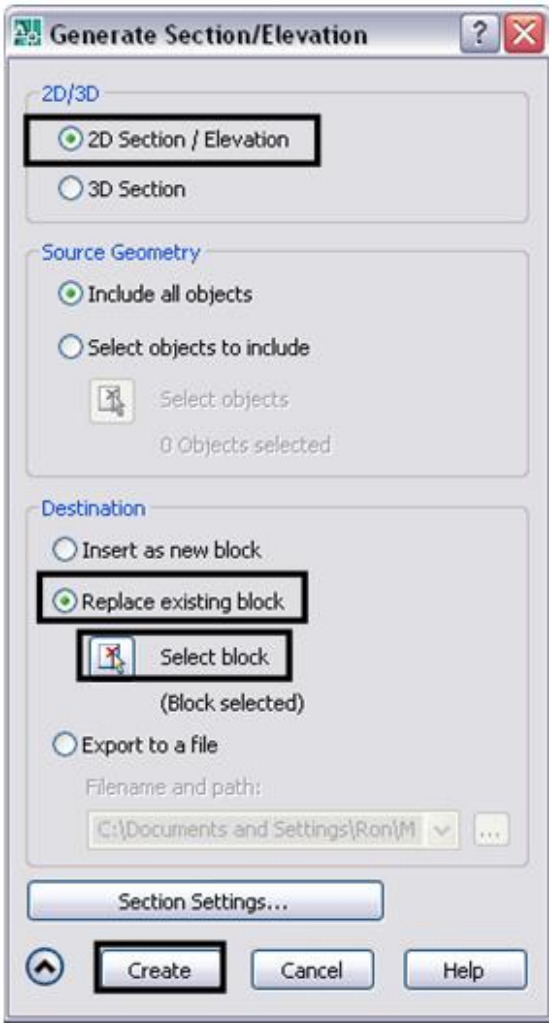
Generating 2D Section

APPLICATION Right Click on section Plan → Generate 2D/3D Sections→ Open Generate Section Window → Check The 2D Section → Create→ Specify the insert point→ Put The " X "Seale factor→ Put The " Y "Seale factor→put the Rotation Angle → Enter

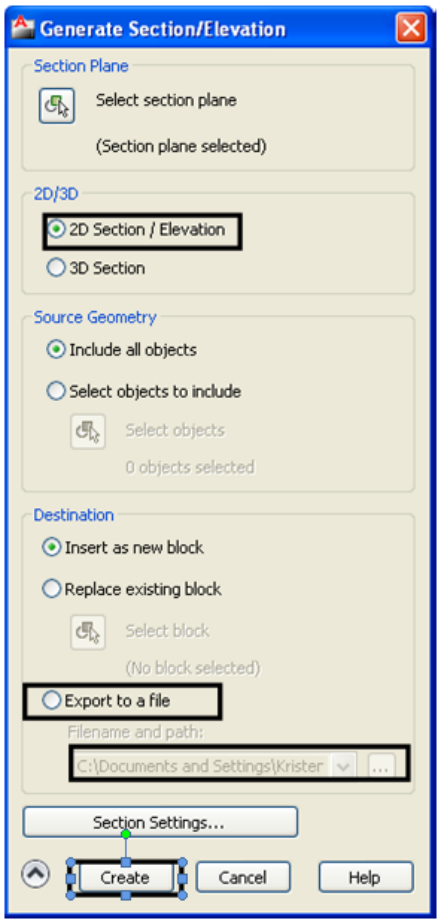
You can insert it as a new block, which is typically what you do the first time to create the section.



Use the replace existing block option to update a block of the section that already exists. You need to click Select Block and select the existing block.

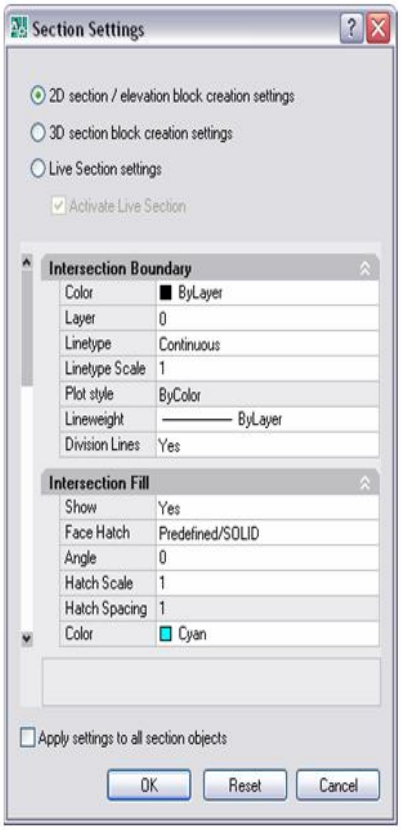


You can also choose to export the section to a separate drawing file by clicking the Export to a File option.



Section Settings

Click Section Settings to modify properties of your section. This button displays the Section Settings dialog.



Use the options in the Section Settings dialog box to control the appearance of objects within your section view. You must first select the appropriate option depending on your current section type. The properties available in the main area of the dialog box depend on the type of section settings you choose

The section settings are stored separately for each section plane. The radio buttons in the upper portion of the dialog determine which settings types are displayed in the properties list.

- **2D Section/Elevation Block Creation Settings**

  Determines how a 2D section or elevation from a 3D object is displayed when generated.

- **3D Section Block Creation Settings**

  Determines how a 3D object is displayed when generated.

- **Live Section Settings**

  Determines how sectioned objects are displayed in the drawing when live sectioning is turned on.

By default, the settings only apply to the current section object. If you select the Apply Settings to All Section Objects check box, then, the settings will apply to all section objects in the drawing.

You can click the reset button to return all settings in the dialog box to their default settings. When you have done all changes, click OK to apply the changes to the section object. Changes will not affect existing blocks until you use the Replace Existing Block tool in the Generate Section/Elevation dialog to update the block

9. Section Command

Use the intersection of a plane and solids to create a region. The SECTION command creates a region object that represents a 2D cross section of 3D objects, including 3D solids, surfaces, and meshes.

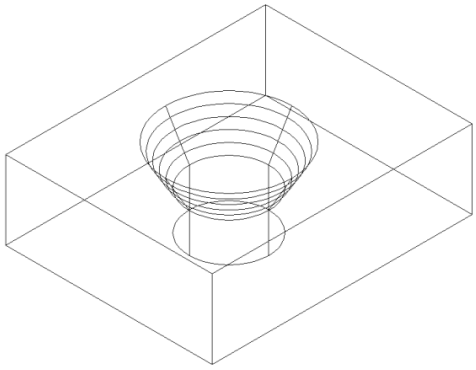
APPLICATION

Type SECTION at the Command Prompt → Select 3D Solid → Enter → Specify first point on Section plane by [Object/Z-axis/View/XY/YZ/ZX/3points] <3points>: → Select first point, Select 2nd Point, Select 3rd point→ to create a 2D region of object.

Section Command Option

1) Object
2) Zaxis
3) View
4) XY
5) YZ
6) ZX
7) 3point

The following prompts are displayed.

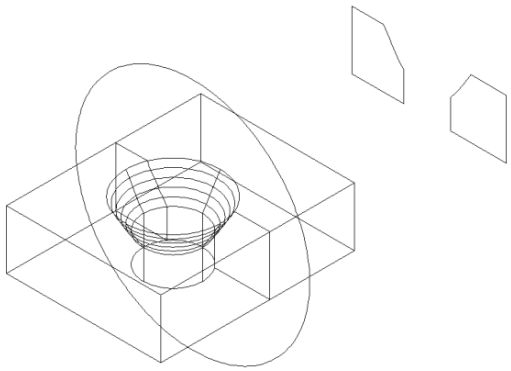


1) Object

Aligns the sectioning plane with a circle, ellipse, circular or elliptical arc, 2D spline, or 2D polyline segment.

APPLICATION

Type SECTION at the Command Prompt → Select 3D Solid → Enter → "O" enter (for Object Option)→ Select sectioning plane →to create a 2D region of object.

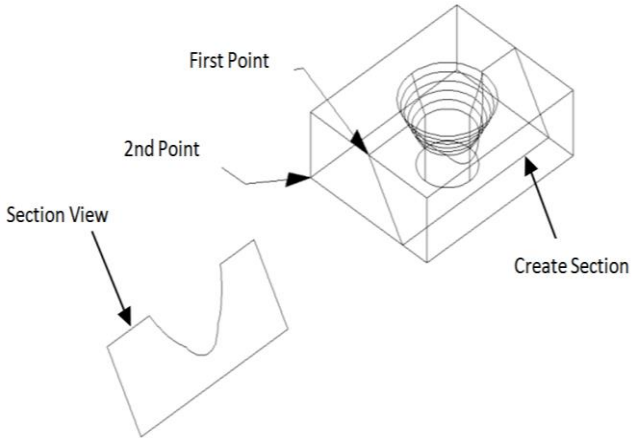


2) Zaxis

Defines the sectioning plane by specifying a point on the sectioning plane and another point on the *Z* axis, or normal, of the plane.

APPLICATION

Type SECTION at the Command Prompt → Select 3D Solid → Enter →"Z" enter (Zaxis Option)→ Select first point on Zaxis → Select 2nd Point on Zaxis→ to create a 2D region of object.



Remember

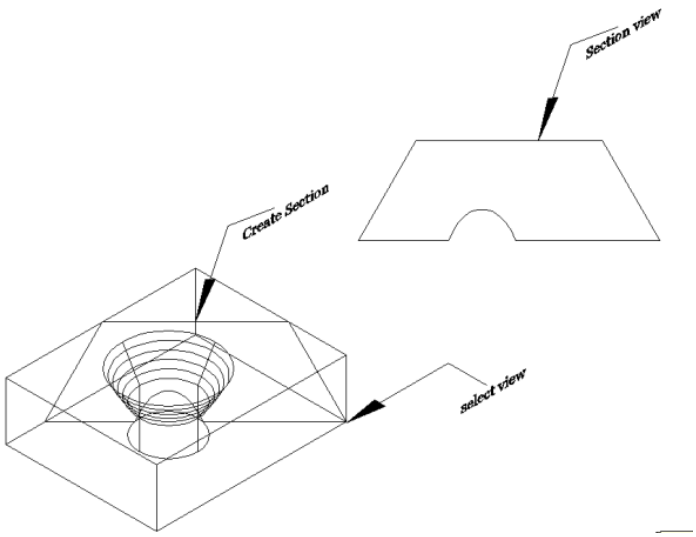
- Point on the section plane. Sets the first point on the plane.
- Point on the Z-axis (normal) of the plane. Specifies a point that defines the axis that is perpendicular to the plane.

3) View

Aligns the sectioning plane with the viewing plane of the current view.

APPLICATION

Type SECTION at the Command Prompt → Select 3D Solid → Enter →"V" enter (For View Option)→ Select View → to create a 2D region of object.

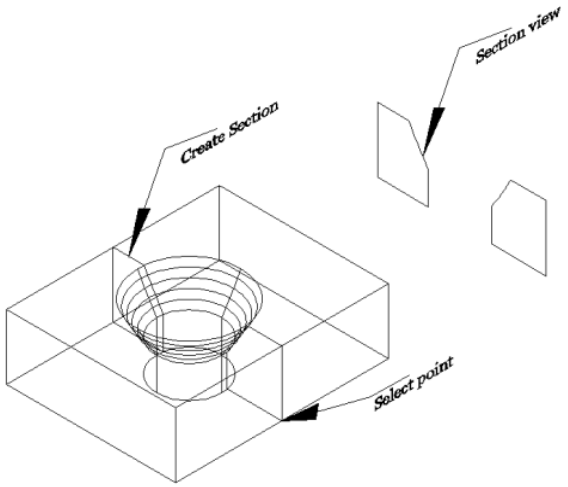


4) XY

Aligns the sectioning plane with the *XY* plane of the current UCS.

APPLICATION

Type SECTION at the Command Prompt → Select 3D Solid → Enter → "XY" enter (For XY plane Option)→ Select point on X axis → To create a 2D region of object.

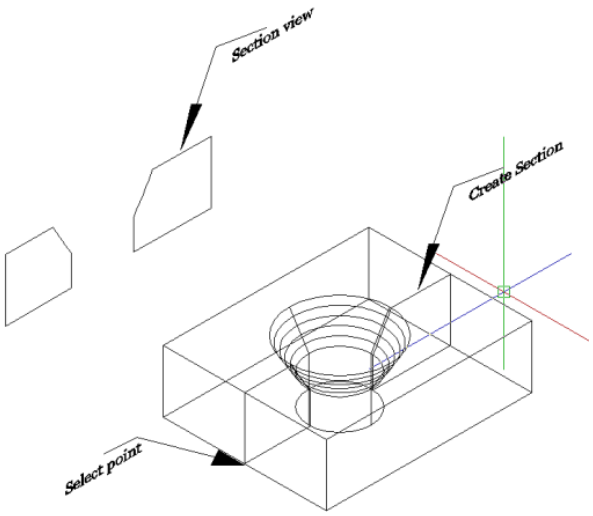


5) YZ

Aligns the sectioning plane with the *YZ* plane of the current UCS.

APPLICATION

Type SECTION at the Command Prompt → Select 3D Solid → Enter → "YZ" enter (For YZ plane Option)→ Select point on Y axis → To create a 2D region of object.



Do you know!

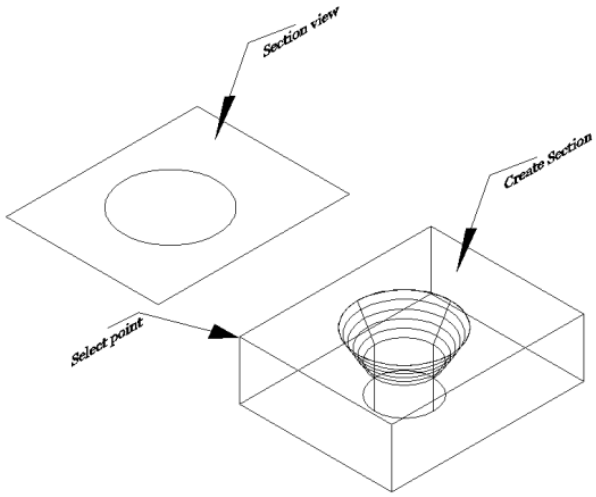
In 3 point, no axis is specify, you can select any 3 points randomly

6) ZX

Align the sectioning plane with the *ZX* plane of the current UCS.

APPLICATION

Type SECTION at the Command Prompt → Select 3D Solid → Enter → "ZX" enter (For ZX plane Option)→ Select point on Z axis → To create a 2D region of object.

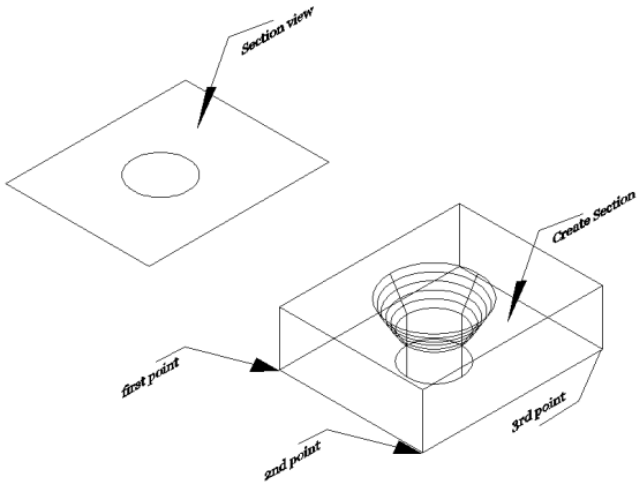


7) 3 Points

Uses three points to define the sectioning plane:

APPLICATION

Type SECTION at the Command Prompt → Select 3D Solid → Enter→ Select first point→ Select 2nd Point→ Select 3rd point→to create a 2D region of object



10. Slice Command

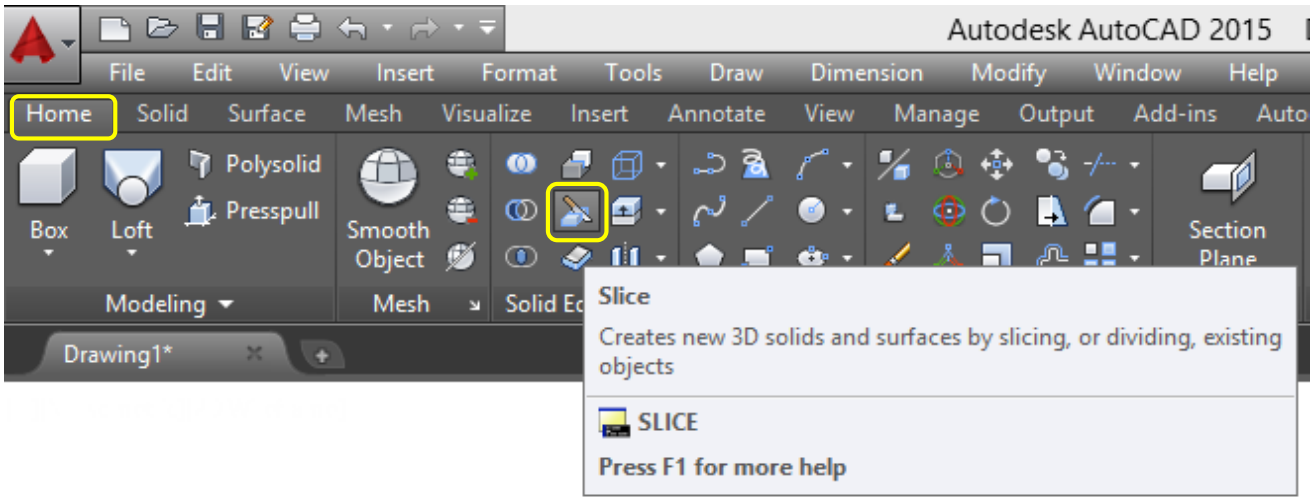
This command does exactly as the name implies. You can slice 3D solid just like you were using a knife. The cutting plane is defined with 2 or 3 points, by specifying a major plane of the UCS, or by selecting a surface object (but not a mesh). Either one or both sides of the sliced 3D solids can be retained.

The sliced objects retain the layer and color properties of the original solids. However, the resulting solid or surface objects do not retain a history of the original objects.

Object that can be used in a slice Operation 3D solid can be sliced at any location by using existing objects such as circles, arcs, ellipses, 2D polylines, 2D splines, or surfaces. Additionally, you can specify a slicing line by picking two points or specify a slicing plane by picking three points. After slicing the solid, you can choose to retain either or both sides of the model. The slices can then be used for model construction or display and presentation purposes. The SLICE command is used to slice solids. When the command is initiated, you are asked to select the solids to be sliced. Select the objects and press [Enter]. Next, you must define the slicing path. The default method of defining a path requires you to specify two points on a slicing plane. The plane passes through the two points and is perpendicular to the XY plane of the current UCS

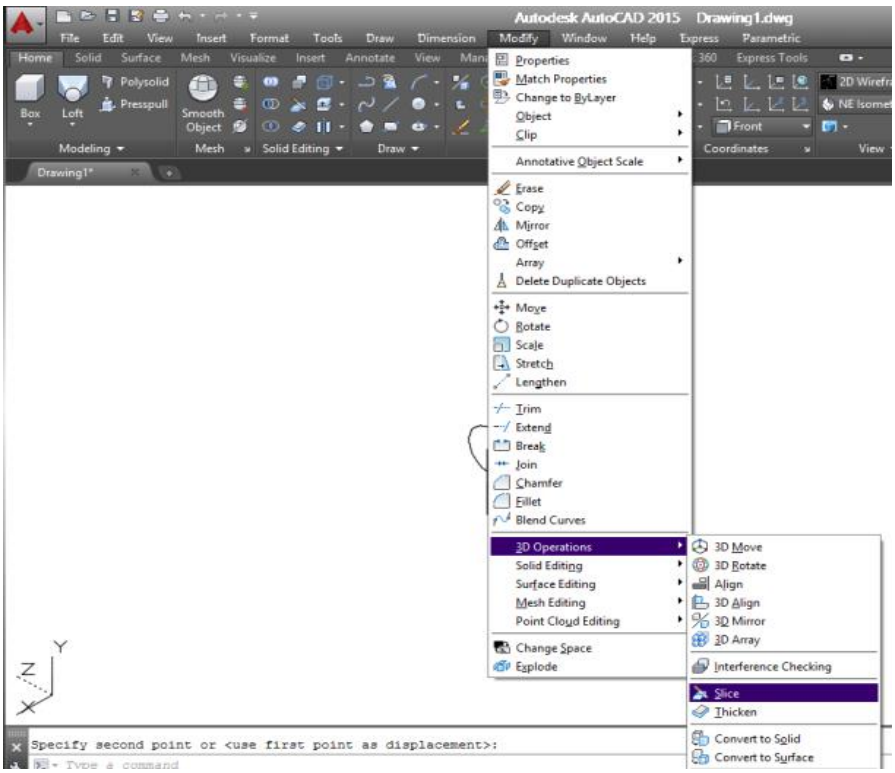
Slice command of 3D solid can be find out through both methods.

Through Ribbon



Through Menu Bar

Menu Bar →3D Operations →Slice



APPLICATION

Type “SL” at the Command Prompt → Select 3D Solid → Enter → Specify first point on Slicing plane or [Planar object/Surface/Z-axis/View/XY/YZ/ZX/3points] <3points>: → Select Fist point on plan ,Select 2nd Point on plan, Specify point on desired side OR [keep Both sides] → B enter.

- Point on desired side
- Keep both sides

Point on desired side

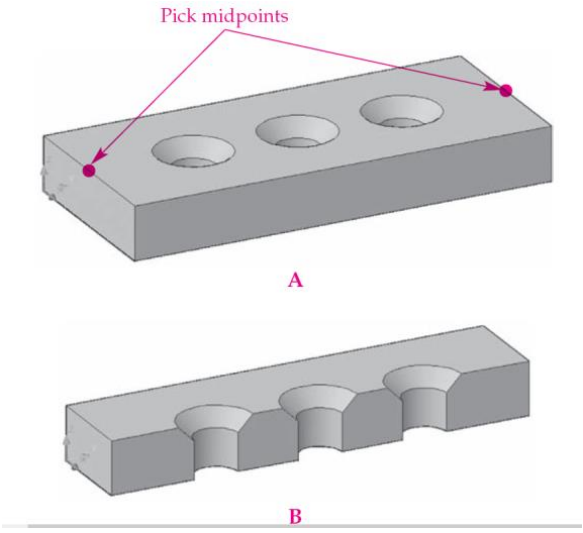
Uses a point to determine which side of the sliced solids your drawing retains. The point cannot lie on the cutting plane.

Keep both sides

Retains both sides of the sliced solids. Slicing a single solid into two pieces creates two solids from the pieces on either side of the plane.

Do you know!

SLICE never creates more than two new composite solids for each selected solid.



Slicing Option

1) Planar object
2) Surface
3) Z-axis
4) View
5) XY
6) YZ
7) ZX
8) 3points

1) Planar Object

A second method to create a slice through 3D solid is the use of an existing planar object. Planar objects include circles, arcs, ellipses, 2D polylines, and 2D splines. The plane on which the planar object lies must intersect the object to be sliced.

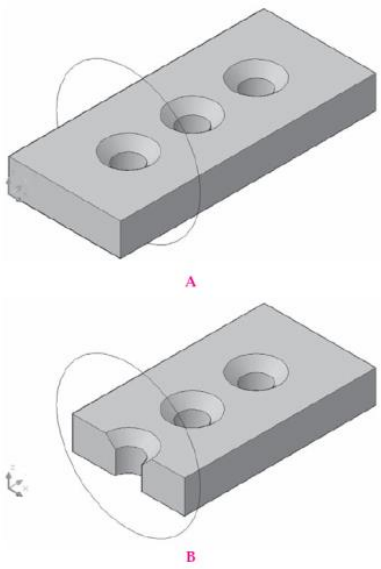
APPLICATION

Type “SL” at the Command Prompt → Select 3D Solid → Enter → “O” enter (For planar object option) → Select Planar objects include circles, arcs, ellipses, 2D polylines, and 2D sp lines. → Desired side OR [keep both sides] → B enter.

Remember

The current UCS has no effect on this option.

- Point on desired side
- Keep both sides



2) Surface

A surface object can be used as the slicing path. The surface can be planar or non-planar (curved). This method can be used to quickly create a mating die.

- First, draw the required surface. The surface should exactly match the stamped part that will be manufactured, Then, draw a box that encompasses the surface.

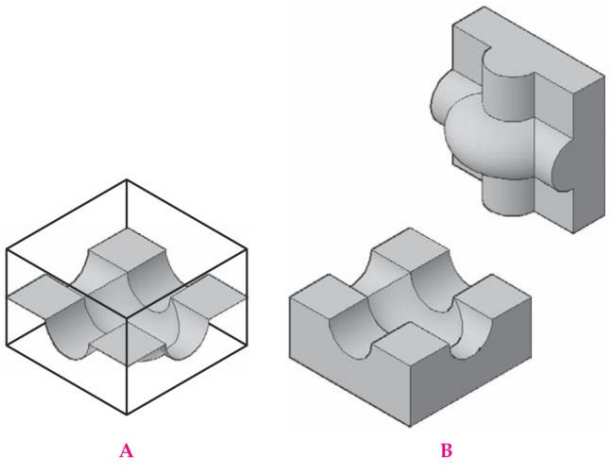
APPLICATION

Type “SL” at the Command Prompt → Select 3D Solid → Enter → “S” enter (For Surface option)→ Select surface → Desired side OR [keep both sides] → B enter.

- Point on desired side
- Keep both sides

Do you know!

You cannot select meshes created with the **EDGESURF**, **REVSURF**, **RULESURF**, and **TABSURF** commands.



3) Zaxis

You can specify one point on the cutting plane and one point on the Z axis of the plane. This allows you to have a cutting plane that is not parallel to the current UCS XY plane.

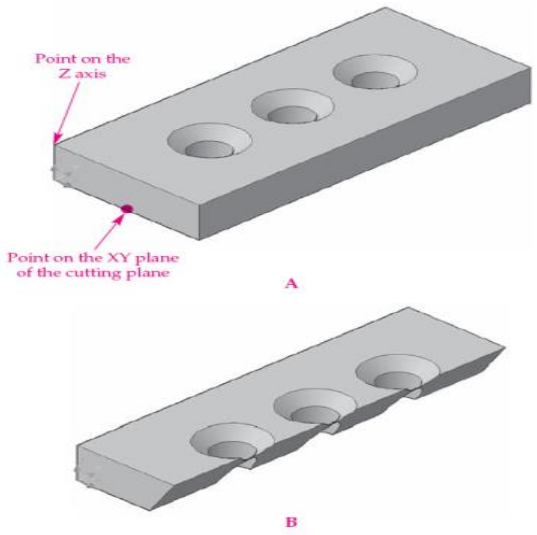
APPLICATION

Type “SL” at the Command Prompt → Select 3D Solid → Enter →”Z” enter (Zaxis Option)→ Select first point on Zaxis → Select 2nd Point on Zaxis→ Desired side OR [keep both sides] → B enter.

- Point on desired side
- Keep both sides

Point to be Noted

- Specify a point on the section plane. Set a point on the slicing plane.
- Specify a point on the Z-axis (normal) of the plane. Specify a point that defines the axis that is perpendicular to the slicing plane.



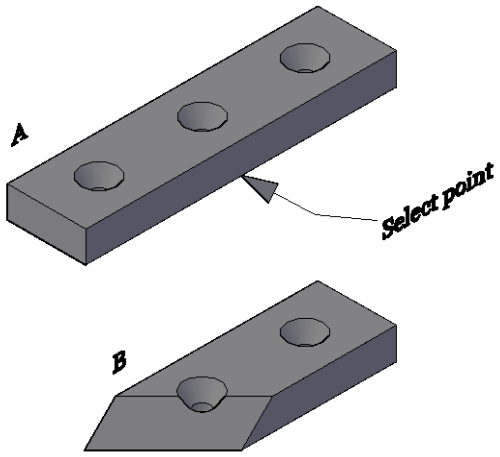
4) View

A cutting plane can be established that is aligned with the viewing plane of the current viewport. The cutting plane passes through a point you select, which sets the depth along the Z axis of the current viewing plane.

APPLICATION

Type “SL” at the Command Prompt → Select 3D Solid → Enter →”V” enter (For View Option)→ Select point on current View → Desired side OR [keep both sides] → B enter.

- Point on desired side
- Keep both sides



5) XY

You can slice an object using a cutting plane that is parallel to XY planes of the current UCS. The cutting plane passes through the point you select and is aligned with the primary plane of the current UCS that you specify.

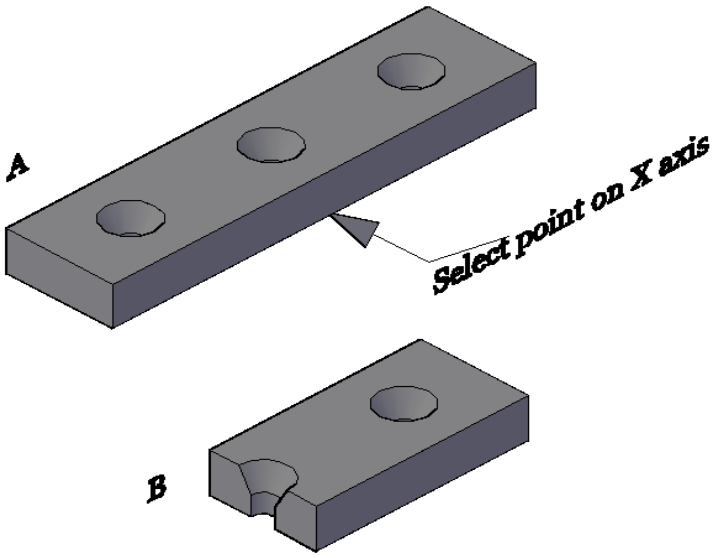
APPLICATION

Type “SL” at the Command Prompt → Select 3D Solid → Enter → “XY” enter (For XY plane Option)→ Select point on X axis → Desired side OR [keep both sides] → B enter.

- Point on desired side
- Keep both sides

Remember

The UCS origin is in the center of the first hole and at the midpoint of the height.



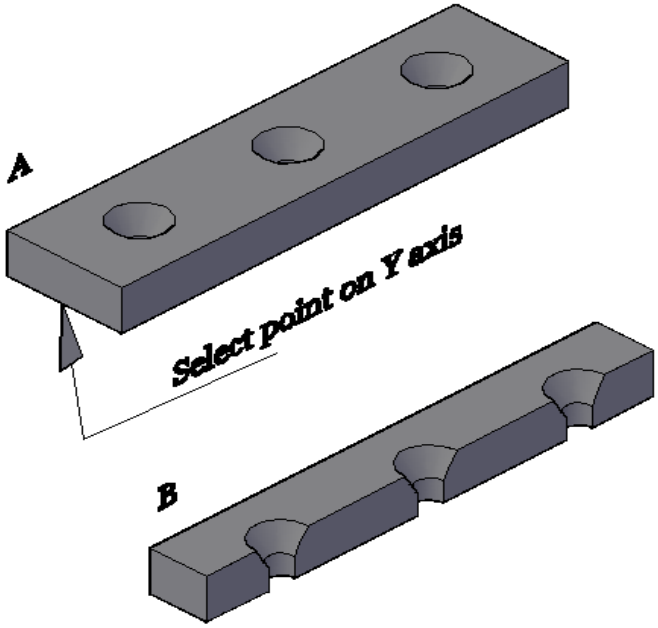
6) YZ

You can slice an object using a cutting plane that is parallel to YZ plane of the current UCS. The cutting plane passes through the point you select and is aligned with the primary plane of the current UCS that you specify.

APPLICATION

Type “SL” at the Command Prompt → Select 3D Solid → Enter → “YZ” enter (For YZ plane Option)→ Select point on Y axis → Desired side OR [keep both sides] → B enter.

- Point on desired side
- Keep both sides



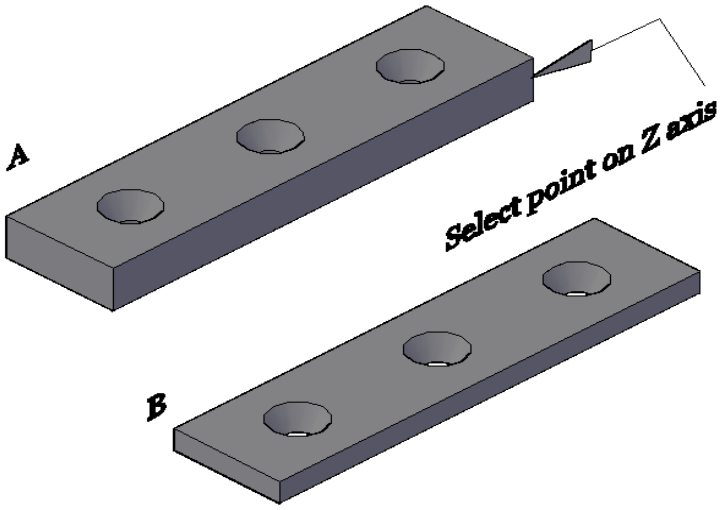
7) ZX

You can slice an object using a cutting plane that is parallel to ZX plane of the current UCS. The cutting plane passes through the point you select and is aligned with the primary plane of the current UCS that you specify.

APPLICATION

Type “SL” at the Command Prompt → Select 3D Solid → Enter → “ZX” enter (For ZX plane Option)→ Select point on Z axis → Desired side OR [keep both sides] → B enter.

- Point on desired side
- Keep both sides



8) Three Points

Three points can be used to define the cutting plane. This allows the cutting plane to be aligned at any angle.

APPLICATION

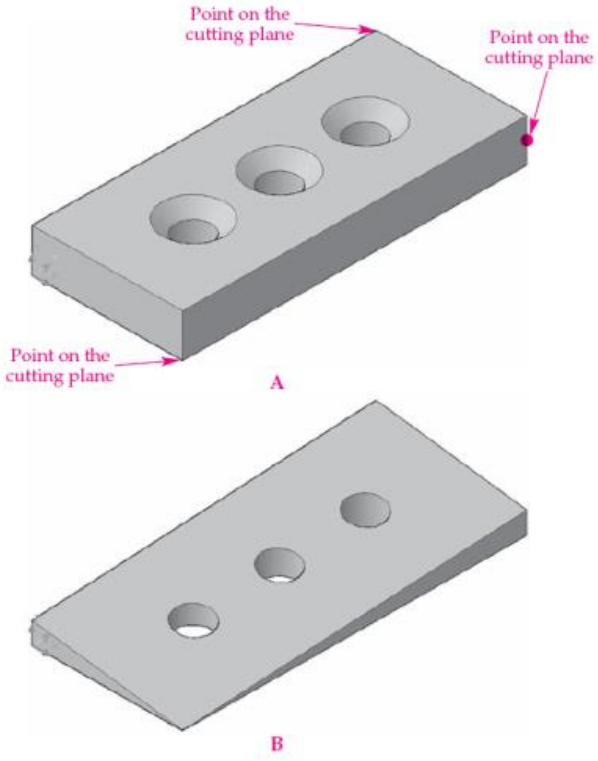
Type “SL” at the Command Prompt → Select 3D Solid → Enter → Select first point→ Select 2nd point → Select 3rd point → Desired side OR [keep both sides] B enter.

- Point on desired side
- Keep both sides

Remember

3 points

3 point can be used at any point on the surface.



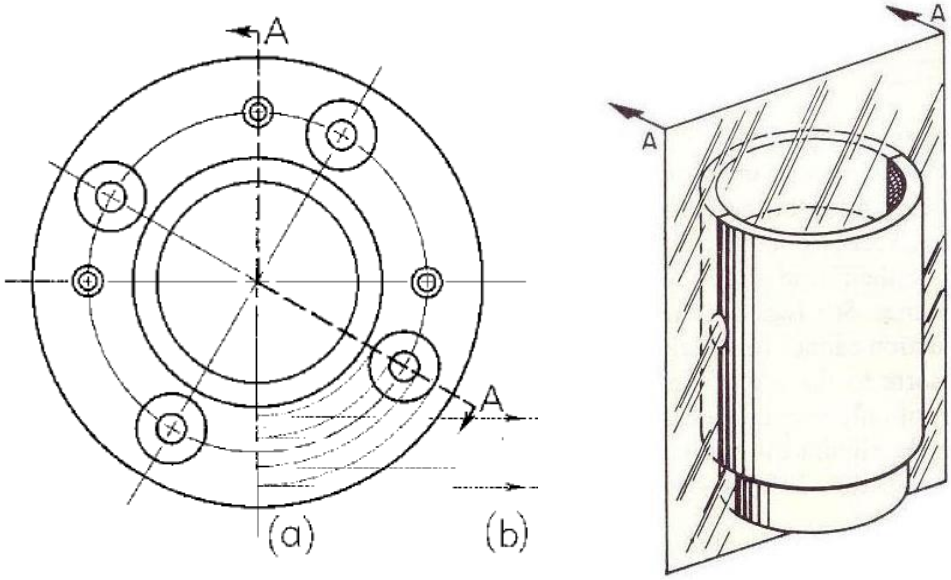
Learner's Activity

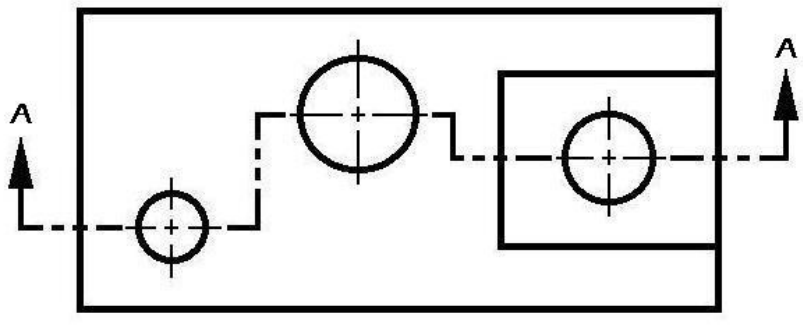
1. **Use the pervious module to generate:**

(For Civil Department)

- 3D section along mid of Y-axis through section plane Command
- Cut the mid of X-axis through slice command

2. **Apply different section commands along the points mentioned in the diagram**
   (For Mechanical Department)





OFFSET SECTION LINE

Learning Unit 2: Merge Flat Objects

Overview

These units describe, how to merge flat objects from 3D model. The following lessons teach you the methods, commands, and options for creating views of your 3D models. These methods include defining views based on orientation of 3D models, planes, and 2D geometry extracted from the 3D models. You will also learn to create, edit, and manage layouts, and combine defining views to create layouts that document your 3D designs.

After completion of this unit you will be able to:

- Extract geometry from solid models
- Use flat shot to create 2D views from your 3D solids
- Create and manipulate section planes
- Create 2D sections or elevations and 3D sections of your 3D solid parts and assemblies
- Create layouts of 3D models including annotation and dimensions

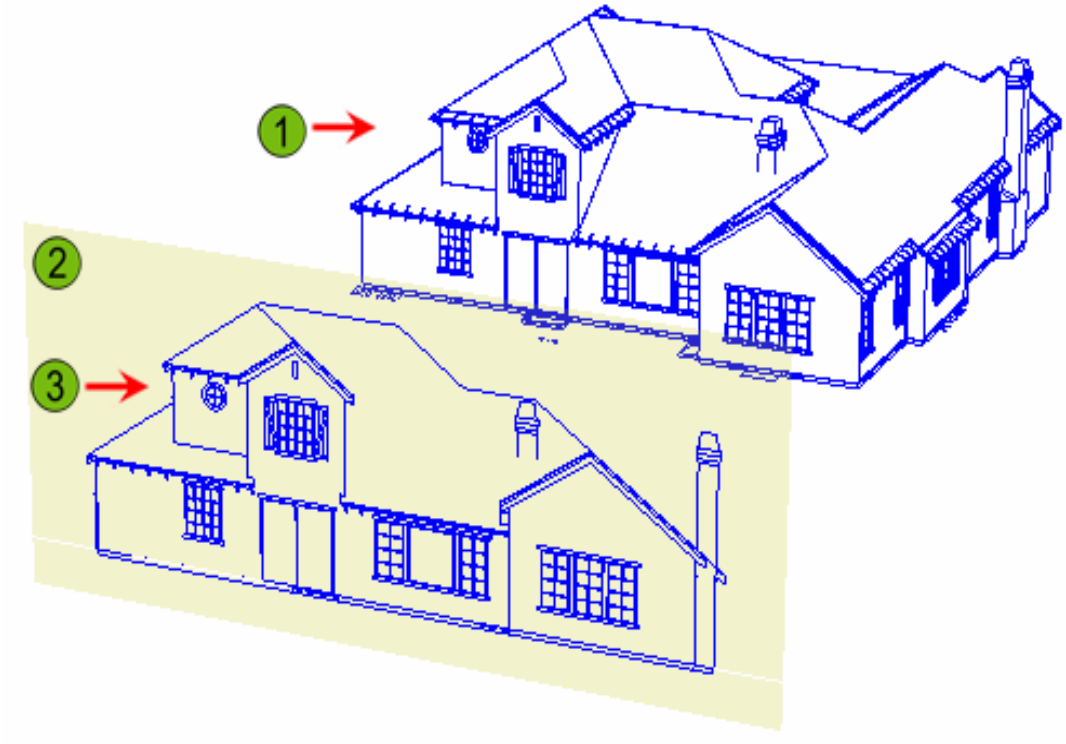
1. Flattened View of 3D Model

Flattened views are generally required when you need to create dimensioned drawings of 3D models. While the 3D model is the closest representation of the finished product, it does not work well for placing dimensions and other annotations for production drawings or construction documentation.

Regardless of the design discipline, 2D geometry is still relied upon heavily when it comes to final documentation and annotation. To make the most use out of your 3D models, you need to create flattened views from those models. In the following illustration, while the 3D perspective image does a better job of conveying the overall design, the 2D elevation view is better suited for construction documentation.

What is a Flattened View?

A flattened view can be defined as a view created from a 3D model in which objects are projected in their current position, relative to the view, onto a plane that is parallel to the current view. Flattened views can be created from any angle and are not limited to standard orthographic projections. In most cases, it is equally beneficial to include a flattened view of an isometric projection. The following illustration shows how a 3D model is projected onto a viewing plane, resulting in a 2D Elevation.



The numbering used in the above diagram describe the following terms:

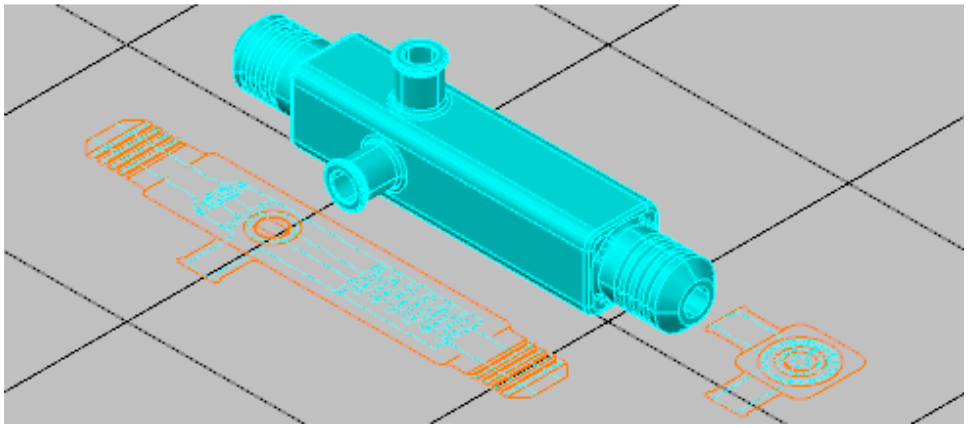
1. 3D model
2. Plane representing 2D view
3. Resulting flattened view for the elevation

Displaying 2D Geometry from a Model View

In this section of the lesson, you will learn how to access the flat shot dialog box and use the options within it to create 2D geometry based on your 3D models. The ability to create accurate 2D data efficiently is significant as the majority of today's industry still relies on 2D data as the primary means of communication between design and manufacturing.

In an architectural environment, flat shot view and drawing can be used to quickly create plan and elevation views to send to clients for review without having to send the entire drawing. DWFs of the flat shot views can be generated for Web-based applications.

In the following illustration, the flat shot command was used to create different 2D views of the 3D model. The display of the 2D geometry is controlled by settings in the Flatshot dialog box.



2. Flatshot Command

Flatshot tool (FLATSHOT command) is used to create a two-dimensional representation of a 3D object. When you use this tool, AutoCAD projects the edges of all 3D solids, surfaces, and meshes in the drawing onto a plane parallel to the current viewing plane. The 2D representation of these edges can then be inserted as a block on the XY plane of the current UCS or saved as a separate drawing.

The process is like taking a photograph of the entire 3D model and then laying the photograph flat. This feature is useful for creating technical illustrations, and can also be used to create two-dimensional elevations.

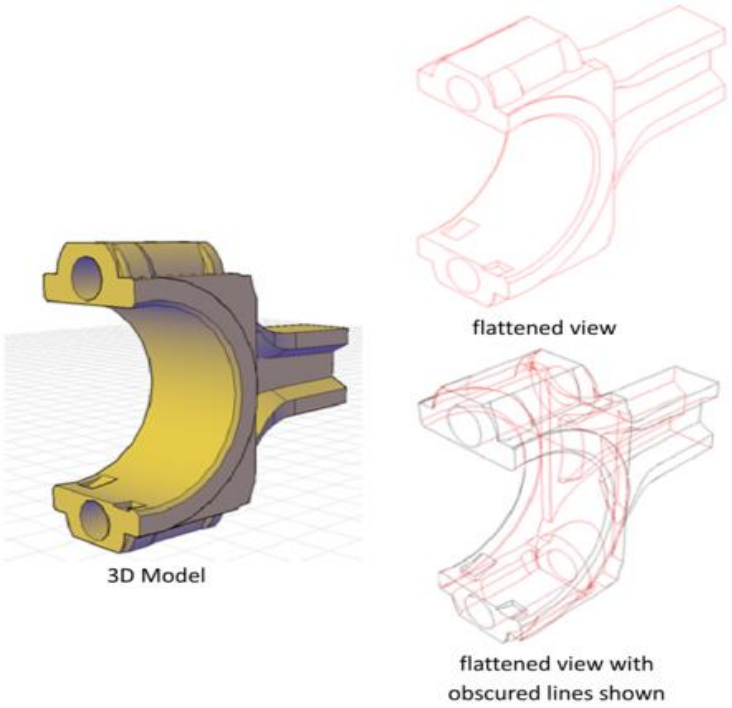
Creating 2D Geometry from a Model View

The Flatshot process works only in model space. Start by setting up the view you want, including orthographic, perspective, or parallel views. All 3D objects in the model space viewport are captured. Therefore, you need to be sure to place the objects you do not want captured onto layers that are turned off or frozen before capturing the geometry.

Once the model is set up the way you want to be captured, select the Flatshot tool from the extended Section panel on either the Home or Mesh Modeling ribbons. AutoCAD displays the Flatshot dialog. This dialog lets you control various aspects for the creation of the 2D representation.

APPLICATION

Type “Flatshot” at the Command Prompt → Open the Flatshot window→ check insert new block option →change the foreground & obscured lines→ Click the create button →click on drawing sheet →enter “X” scale factor →put the rotation angle.



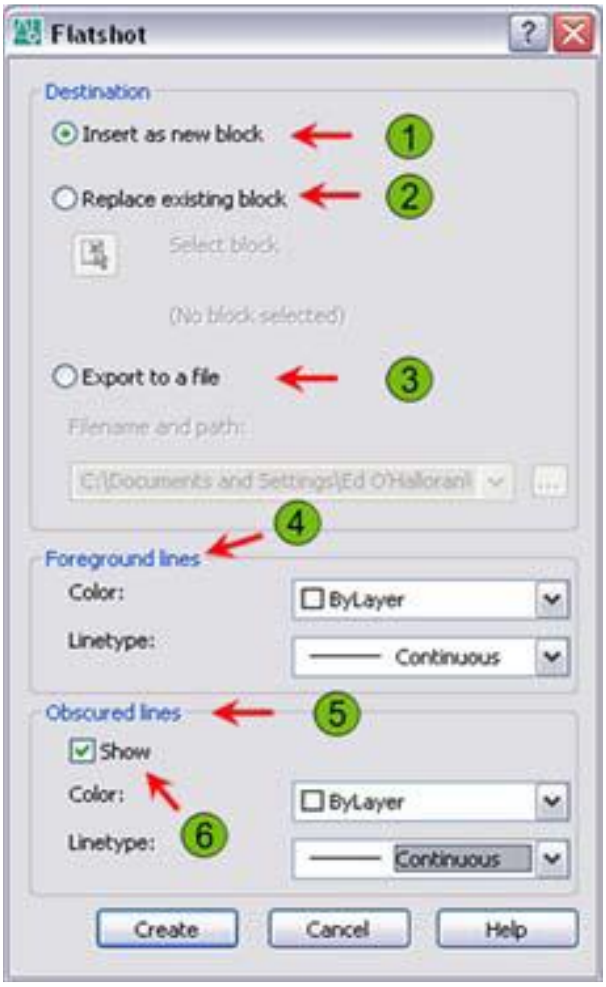
Do you know!

You can turn off or frozen the object that you do not want to capture in

Options of Flatshot command:

- **Base point** Use to specify the insertion point of the geometry.
- **Scale** Use to scale the geometry uniformly in all axes.
- **X** Use to scale the geometry in the X direction independently of the Y and Zaxis.
- **Y** Use to scale the geometry in the Y direction independently of the X and Zaxis.
- **Z** Use to scale the geometry in the Z direction independently of the X and Yaxis.
- **Rotate** Use to rotate the geometry on the current XY plane.

Flatshot Dialog Box



Step-1: Insert as New Block:

Use to create 2D geometry of the 3D model as a block.

Step-2: Replace Existing Block:

Use to update an existing view to reflect changes in the solid model.

Step-3: Export to a File:

Export the 2D geometry to be available for use in other drawings.

Step-4: Foreground Lines:

Use to set the color and line type of the visible lines.

Step-5: Obscured Lines:

Use to set the color and line type of hidden lines.

Step-6: Show:

Use to toggle the display of hidden

3. Review Flatshot Dialog Box

1) Destination

Controls where the flattened representation is created.

a) Insert As New Block

Specifies to insert the flattened representation as a block in the current drawing.

b) Replace Existing Block

Replaces an existing block in the drawing with the newly created block.

c) Select Block

Closes the dialog box temporarily while you select the block you are replacing in the drawing. When you finish selecting the block, press Enter to redisplay the Flatshot dialog box.

d) Block Selected / No Block Selected

Indicates whether a block has been selected or not.

e) Export to a File

Saves the block to an external file.

2) Foreground Lines

Contains control for setting the color and types of lines that are not obscured in the flattened view.

a) Color

Sets the colors of lines that are not obscured in the flattened view.

b) Line type

Sets the types of lines that are not obscured in the flattened view.

3) Obscured Lines

Controls whether lines that are obscured in the drawing are displayed in the flattened view, and sets the color and line type of these obscured lines.

a) Show

Controls whether obscured lines are shown in the flattened representation. When selected, the 2D flattened representation displays lines hidden by other objects.

b) Color

Sets the colors of lines that lie behind geometry in the flattened view.

c) Line type

Sets the line types of lines that lie behind geometry in the flattened view.

4) Include Tangential Lines

Creates silhouette edges for curved surfaces.

5) Create

Finally creates the flattened view.

Using Flatshot to Create 2D Elevations

Remember

You need to be sure to place the objects you do not want captured onto layers that are turned off or frozen before capturing the geometry.

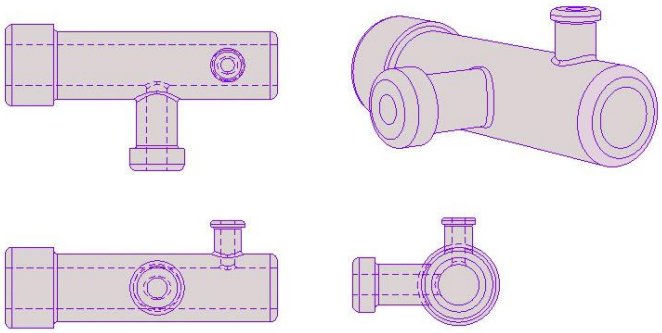
While the Flatshot tool is most useful for creating two-dimensional representations of 3D models shown in a perspective projection to depict the three-dimensional geometry for creating technical illustrations, if you first reorient the model to display a parallel projection elevation view, this tool can also be used to create two-dimensional elevation views.

An unnamed block is inserted consisting of the flattened geometry. The block is inserted on the current layer. Because the block is aligned with the current UCS, it may appear strange when first inserted. However, if you reorient the drawing to a plan view of the current UCS, the elevation will appear correct

1. Use the View Cube to reorient the model to the desired top, front, or side orthographic view.
2. Change the projection mode to a Parallel projection

APPLICATION

Type “Flatshot” at the Command Prompt à Open the Flatshot windowà Check Box Insert as new Block à Adjust the settings for foreground and obscured lines and determine whether to include tangential edges à Click Create à In the drawing, specify an insertion point for the block and press ENTER to accept the default values for scaling and rotation angle



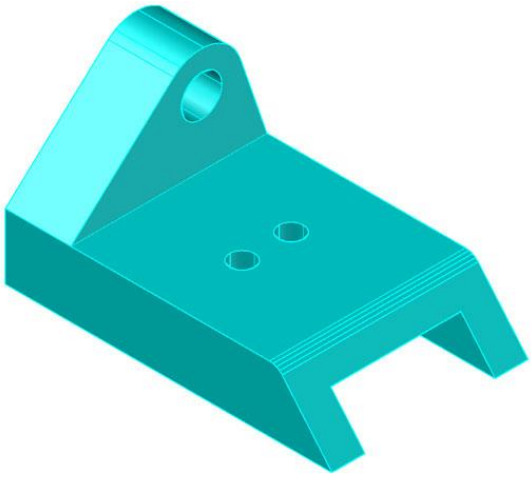
Remember

Create the elevations on the WCS and then use commands such as MOVE to arrange the elevations into the proper positions for a typical set of orthographic views.

4. Practical Application

Let's Take a Flatshot of a Mechanical Object

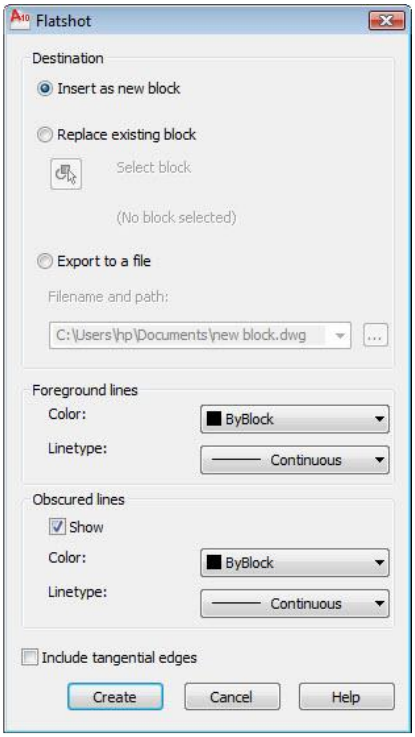
The Flatshot command is the oldest, fastest, easiest to use, and least versatile of the methods for producing 2D output from a 3D model.



Step-1:

First, let's create the top view of the model. Use the View command to align the view of the model so we are looking straight down on top of it [View | Views | Top] or {View | 3D Views | Top}.

Step-2:



Now start the Flatshot command [When in the 3D Modeling workspace, select Home | Section | Flatshot]. This brings up the dialog box below:

Then just accept the defaults as indicated and click Create.

AutoCAD churns for a brief moment, and then starts issuing a series of prompts that look exactly like those from the Command-line version of the Insert command.

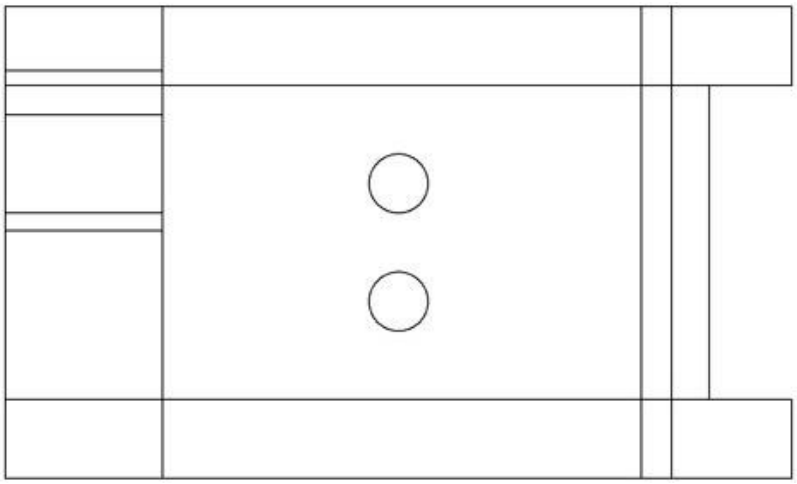
This is not a coincidence, because that is exactly what is happening. AutoCAD has projected all the edges of the model onto the XY plane of the current coordinate system (the WCS in our case), and created lines, circles, arcs, and ellipses of the edges, which it then bundles up into a block definition.

Step-3:

Start by selecting a suitable insertion point. Next, AutoCAD asks for X and Y scale factors, which should normally be 1 by default, because we usually draw everything, full size in AutoCAD. Finally, it asks for a rotation angle:

Take the default value of 0 (zero).

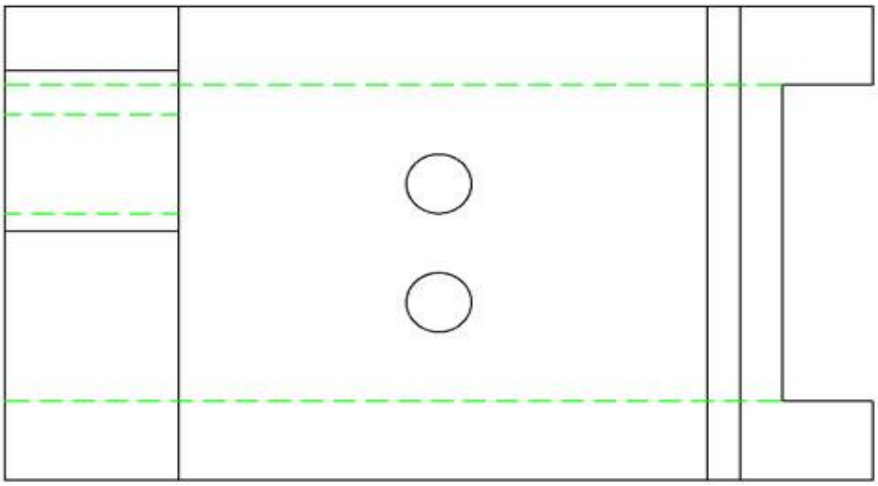
And here we are! The 3D part has been projected into a flat 2D view.



But, that's not quite what we expected. Why are there so many extra lines, and why doesn't it show hidden lines properly?

Step-4:

Okay, let's try this again. Run the Flatshot command again, but this time go to the Obscured Lines window and set the color to Green and the line type to Hidden. Click on the Replace existing block button and select the existing projected top view. Click Create.



Ah, that's better.

Step-5:

Now let's do the front view. Use the View command to change the current view to Front and run Flatshot again.

Step-6:

Once again, AutoCAD is asking for an insertion point for a block insertion, but all we can see is what appears to be a single line. That is correct, because that's how Flatshot works, and this is a significant point. It takes the current screen view direction and projects it onto the XY plane of the current UCS.

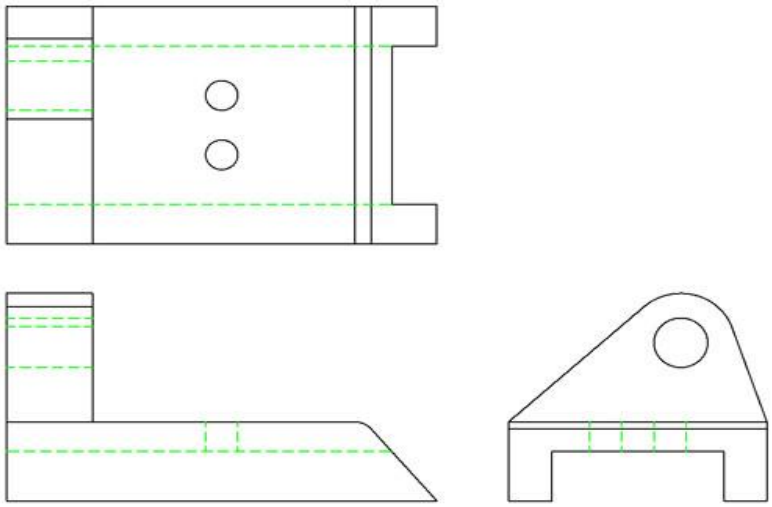
In the case of the top view, it projected the XY plane onto the XY plane. In the case of the front view; however, we are looking at the XZ plane and projecting it onto the XY plane, so we are looking at the edge of the block insertion.

It can be difficult to place a block insertion accurately under these conditions, so the best bet is to type in a coordinate triplet such as 0, 0, and 0.

We should always move the block later.

Step-7:

This time, shift the View to Right and run Flatshot once again now specifying a desired insertion point for the block. Now let's look at the Top view again. After a bit of re-arranging of the block insertions (the quick and easy way is to use parametric *collinear* constraints), we have something that looks like this:



That was easy, wasn't it!

As noted earlier, Flatshot is simple, fast, and easy to use, but not too versatile.

How to create Tangential line?

You may have noticed the Include Tangential Edges button at the bottom of the Flatshot dialog box. When it is turned on, then a line appears at any place that two faces are tangent to each other. For example, look at the right-hand side of our top view and compare it with the front view. The two tangent edges of the fillet have produced the two parallel lines in the top view. If it was off, then these two lines wouldn't appear, and there would be nothing in the top view to indicate the change in direction of the top face.

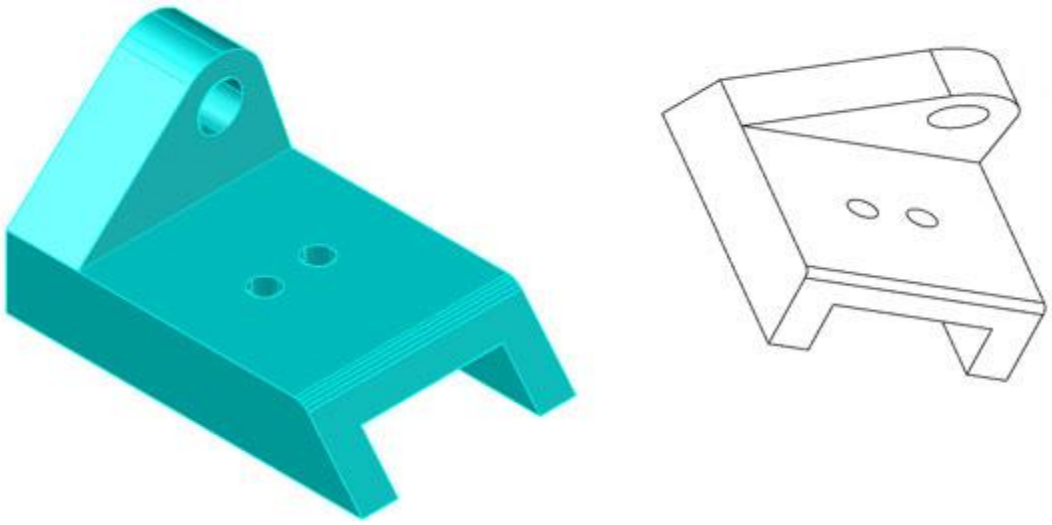
Naming of 2D View

You don't have to be stuck with the long, esoteric name that AutoCAD generates when it creates 2D views. You can rename them to something meaningful, such as Top, Front, and so on.

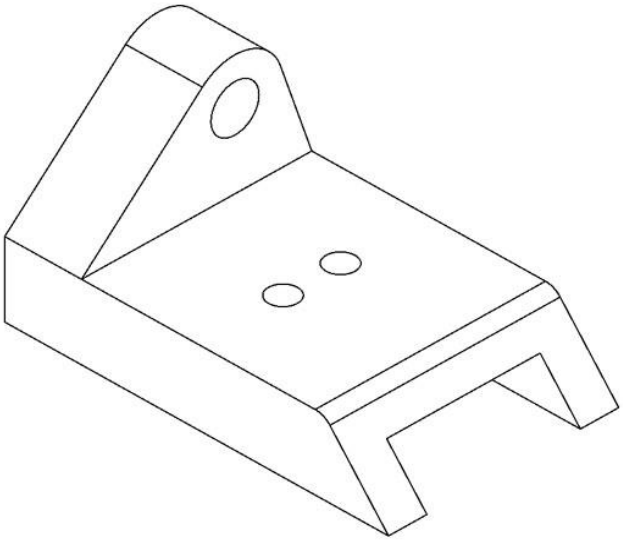
So what happens if you make changes to the original 3D solid model? No problem. Just run Flatshot again, but this time select Replace Existing Block in the Destination window of the Flatshot dialog box. Select the appropriate block (here is where renaming comes in handy), and everything updates.

Isometric View

This can be slightly confusing for the first time you produce one. Simply align the current screen view to an isometric orientation and run Flatshot. The problem would seem to be that what it produces doesn't look like a proper isometric view.



Ah, but don't forget that Flatshot projects onto the XY plane of the current UCS, and we are looking at this from a perspective angle. Flip back to the top view and everything is corrected. This view also contains good examples of the tangent line option.



There is another issue about isometric views. Technically, Flatshot produces an isometric *projection*, whereas standard drafting practice is often to draw an isometric *view*. The difference is that an isometric *view* is normally drawn so that lines that are parallel to the three principle axes appear in their true length, while an isometric *projection* foreshortens them due to the tilting and rotating of the viewing angle of the object.

The solution is simply to ignore our usual rule about drawing and inserting at full size. When inserting an isometric projection, use a scale factor of approximately 1.23 and you will get an isometric view. You can also edit the insertion later to make it match this scale factor.

And Finally

Remember

Always check your units of measure.

There is one final button on the Flatshot dialog box that can be useful. Click on the Export to a file button. You can now browse to a folder location and assign a file name. The resulting extracted view gets written out to a new drawing file. If you create multiple views then they each need their own file, but you can easily Insert them back into a master drawing to create your final multi-view drawing.

Advantages of 2D view

This capability has two major uses:

First, you can extract and send 2D working drawings to clients, customers, and vendors without having to send the basic design information contained in your 3D model.

The other procedure is to build the 3D model, extract the 2D drawing views, and go home for three days. Now come back and submit the drawing to your boss, who will be very impressed with the fact that you managed to create such a complex drawing, including the isometric view, in only three days.

5. Using Solid View, Solid Draw and Solid Profile Tools

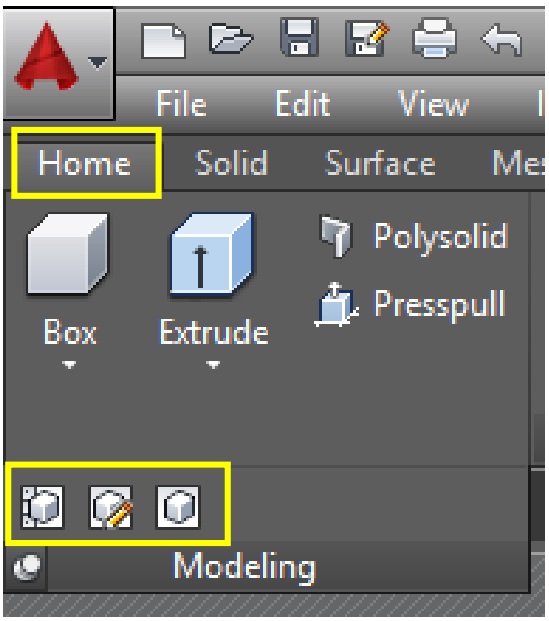
You can also use three related tools to automate the steps needed to create 2D representations of solids that mimic conventional 2D drafting. These three tools

1. **Solid View**
2. **Solid Draw**
3. **Solid Profile**

These tools are found by expanding the Modeling panel on the Home ribbon tab (When working in the 3D Modeling workspace).

Remember

The information stored on these layers is deleted and updated when you run SOLDRAW. Do not place permanent drawing information on these layers



Solid View (SOLVIEW)

This command automates the Manual process of creating views, layers, and layout viewports for 3D models.

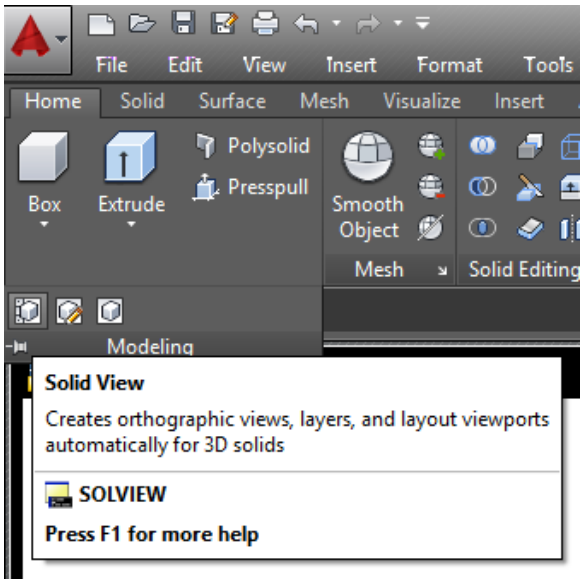
SOLVIEW must be run on a layout tab. If the Model tab is current, the last active layout tab is made current.

SOLVIEW places the viewport objects on the VPORTS layer, which it creates if it does not already exist. The view-specific information that is saved with each of these viewports is used by SOLDRAW to generate the final drawing view.

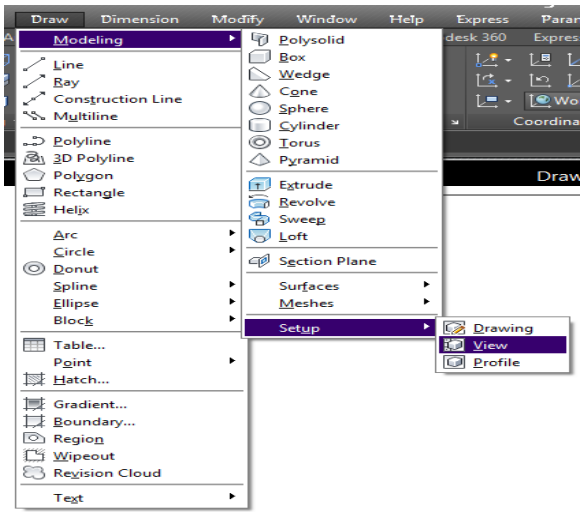
SOLVIEW creates layers that SOLDRAW uses to place the visible lines and hidden lines for each view, view name-VIS, view name-HID, view name-HAT, and a layer where you can place dimensions that are visible in individual viewports, view name-DIM.

Solid View command can be find out through both ways

Through Ribbon



Through Menu Bar

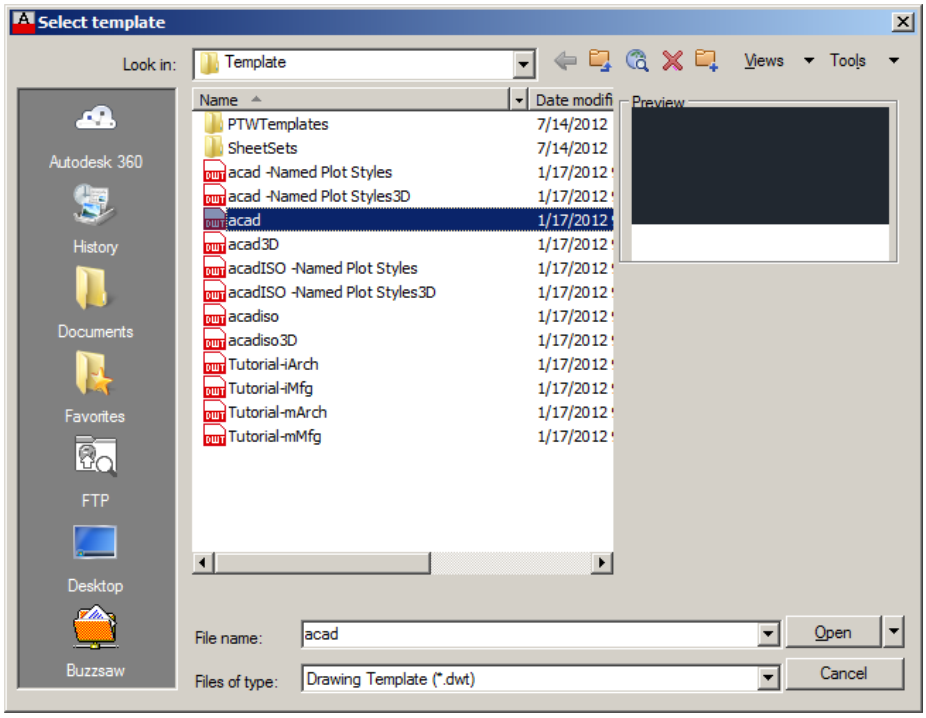


Following are the steps for this command

Step 1: Choose File, New...

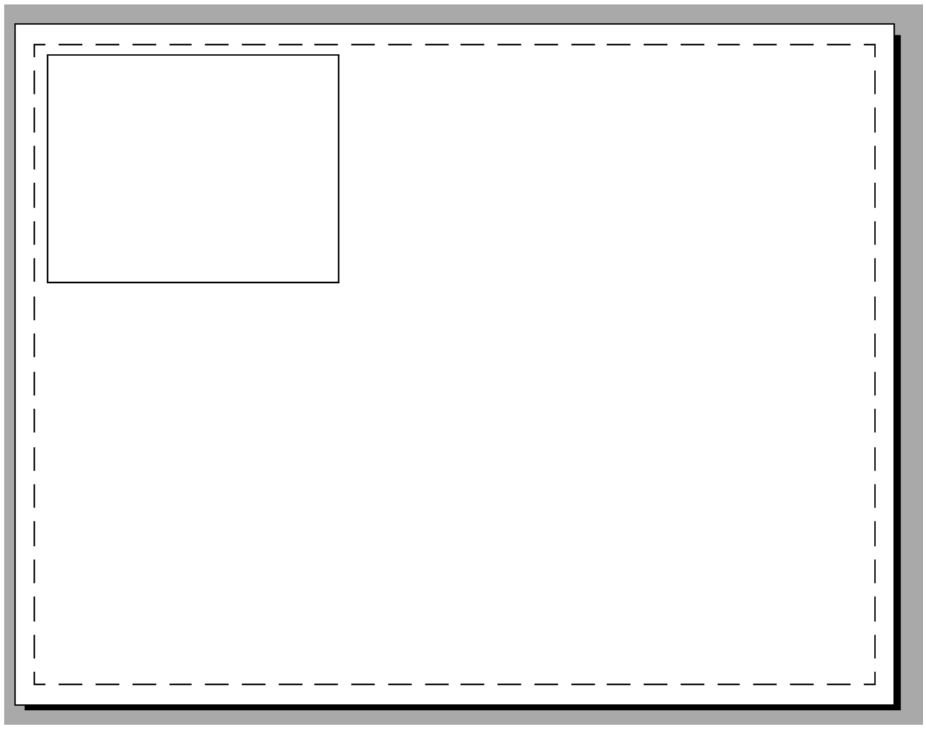
Step 2: Choose “Use a Template” from the startup dialog.

Step 3: Double-click **acad.dwt** as the template file to start from.



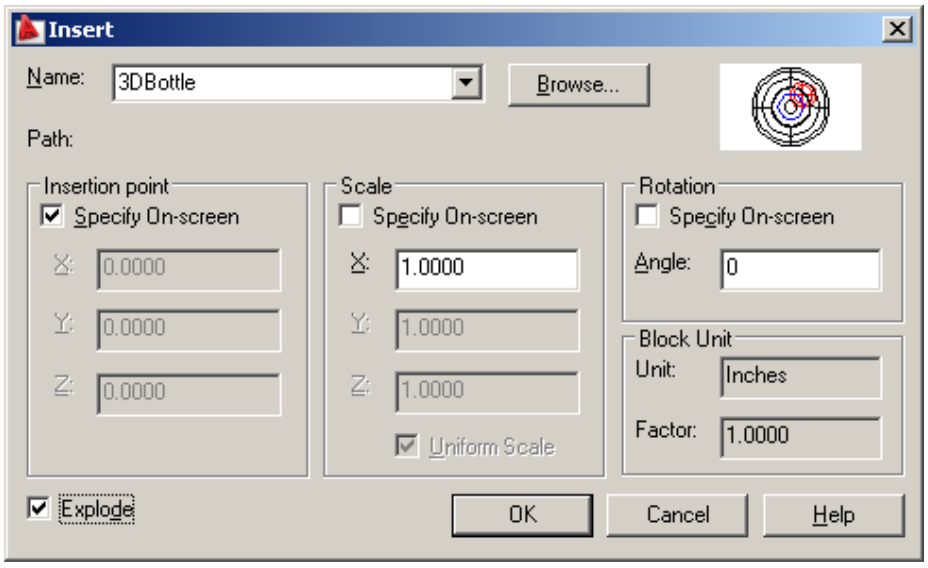
Step 4: Create a new layout.

Step 5: Move and resize the model view so it is in the upper left corner of the sheet as follows.



Step 6: Type MSPACE at the command prompt.

Step 7: Type INSERT at the command prompt and choose the drawing called 3DBOTTLE.DWG to insert. Explode the Block as you are inserting it.



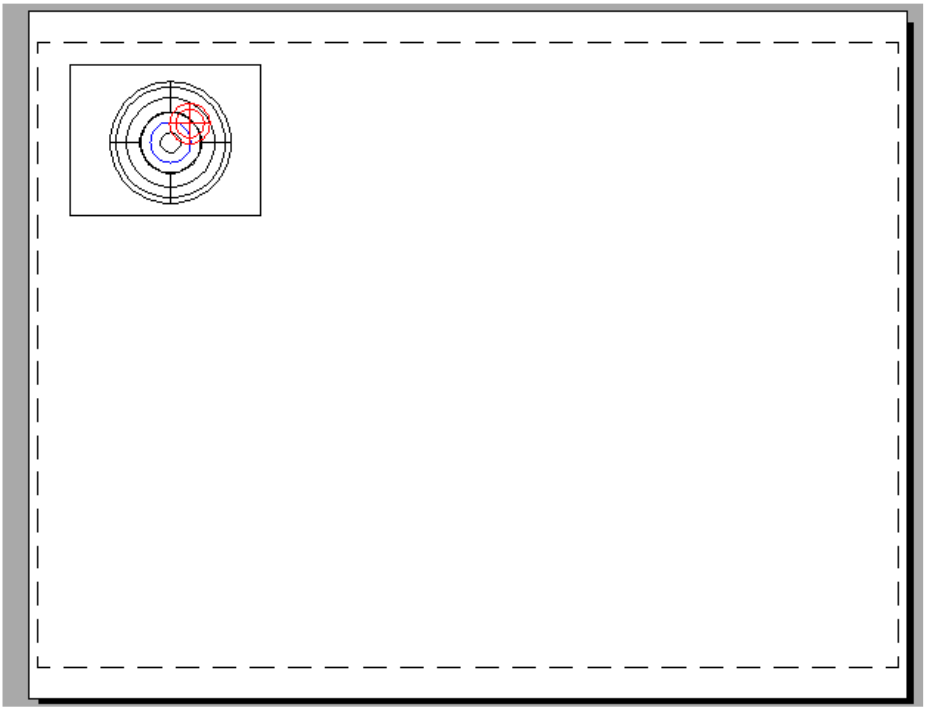
Step 8: Insert the block anywhere in the Model Space view at a scale of 1 and rotation 0.

Step 9: Type ZOOM EXTENTS then ZOOM 1XP.

Step 10: Type PSPACE at the command prompt.

Step 11: Change the layout paper size to ANSIC (22 x17).

Model Space View Scaled 1XP



Step 12: Type **SOLVIEW** at the command prompt.

Command: Solid view (short key=**SOLVIEW**)

Ucs/Ortho/Auxiliary/Section/<exit>: **O** Pick side of viewport to project: **pick bottom "front" of the viewport.**

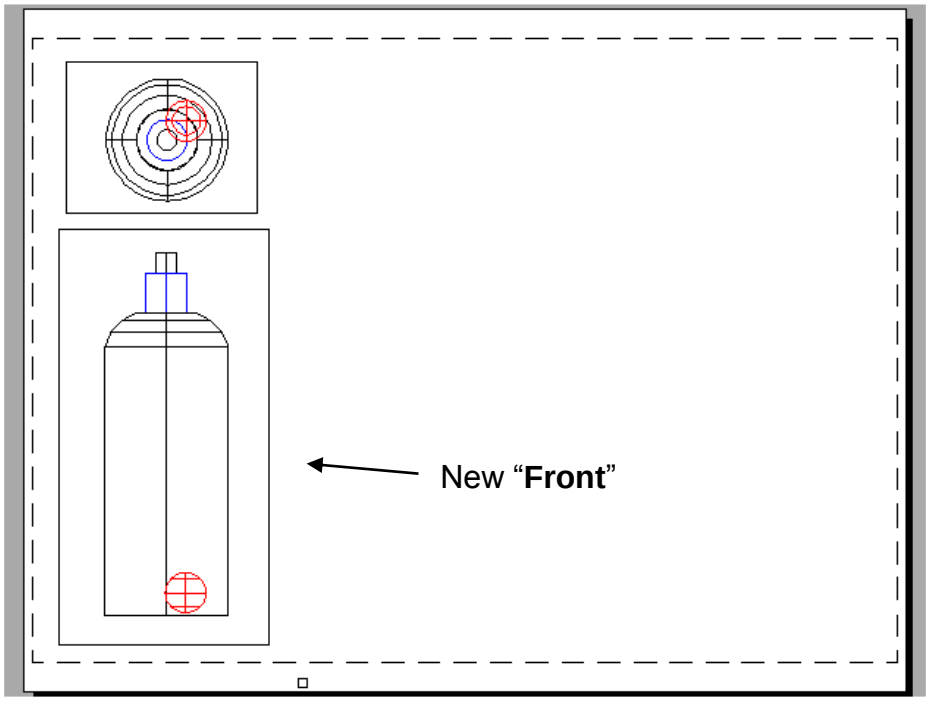
View center: **drag a point below the current viewport**

Specify first corner of viewport: **click upper left corner**

Specify opposite corner of viewport: **click lower right corner**

Enter view name: **FRONT**

Enter an option [Ucs/Ortho/Auxiliary/Section]: **enter**

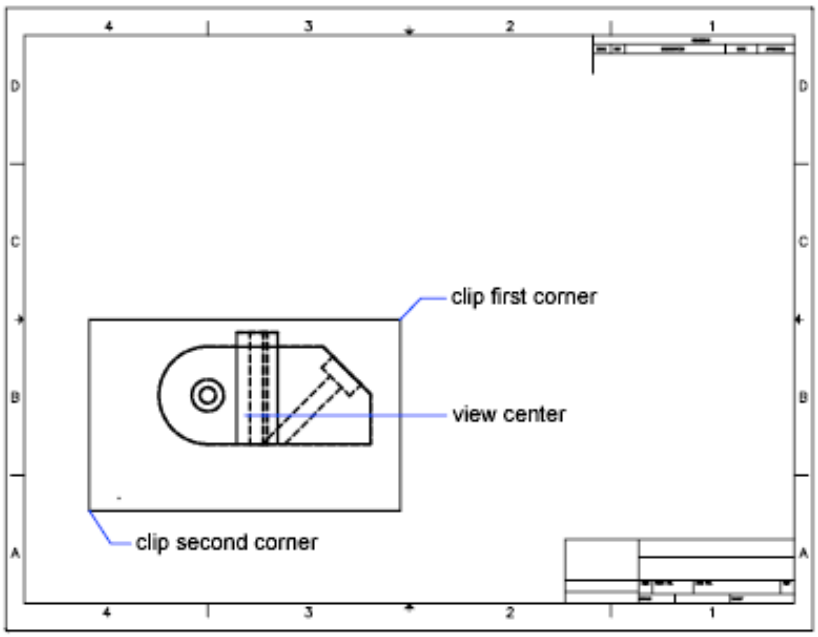


Solid View Options

1. **UCS**
    a) Named
    b) World
    c) List Named UCSs
    d) Current
2. **Ortho**
3. **Auxiliary**
4. **Section**

1) UCS

Create a profile view relative to a user coordinate system. If no viewport exists in your drawing, the UCS option is a good way to create an initial viewport from which other views can be created. All other SOLVIEW options require an existing viewport. You have the option of using the current UCS or a previously saved one as the profile plane. The viewport projection is created parallel to the XY plane of the UCS with the X axis facing right and the Y axis upward.



a) Named

Use the XY plane of a named UCS to create a profile view. Enter the name of the UCS you want to use and the scale of your view. Entering a scale is equivalent to zooming your viewport by a factor relative to paper space. The center is based on the current model space extents.

b) World

Use the XY plane of the WCS to create a profile view. Enter the name of the UCS you want to use and the scale of your view.

c) List Named UCSs

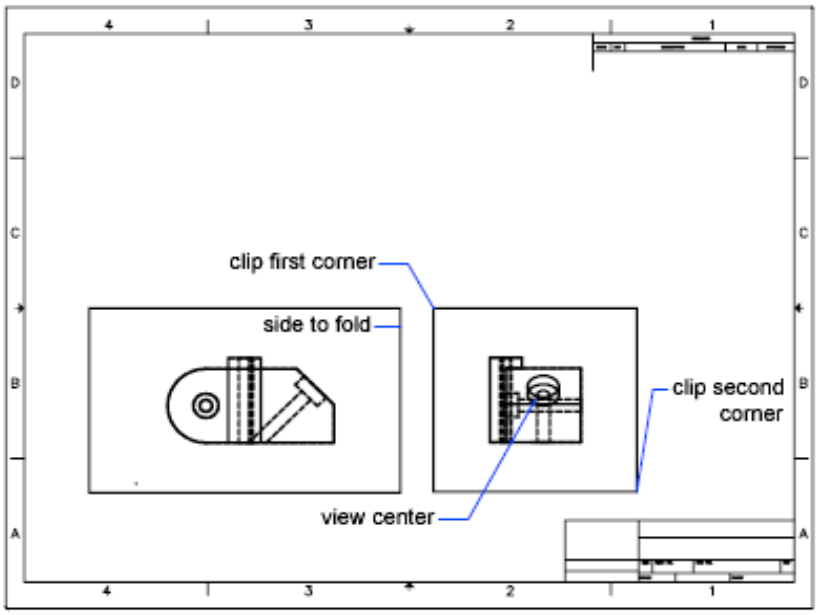
List the names of existing user coordinate systems. The list is filtered using the wild-card combinations you enter (wild-card characters accepted by the UCS command are valid).

d) Current

Use the XY plane of the current UCS to create a profile view.

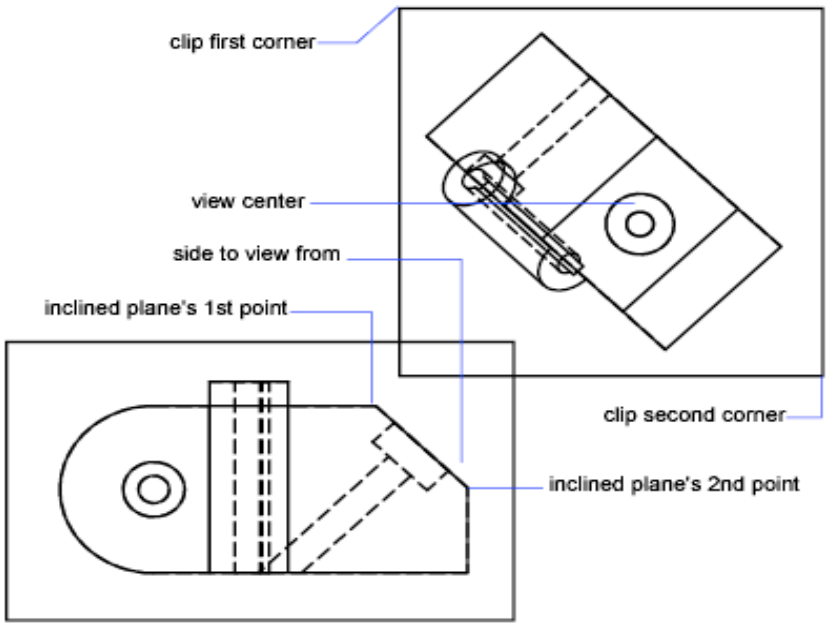
2) Ortho

Create a folded orthographic view from an existing view. Once you select the side of the viewport you want to use for projecting the new view, a rubber-band line perpendicular to the side of the viewport helps you to locate the center of the new view.



3) Auxiliary

Create an auxiliary view from an existing view. An auxiliary view is one that is projected onto a plane perpendicular to one of the orthographic views and inclined in the adjacent view.

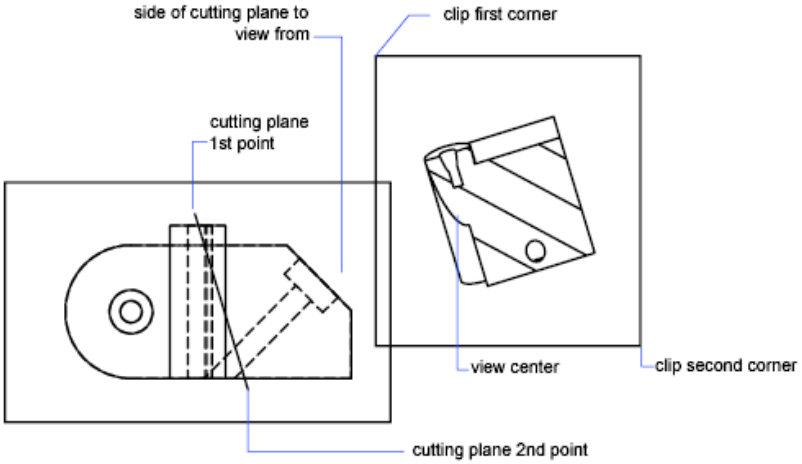


Two points define the inclined plane used for the auxiliary projection. Both points must be located in the same viewport. A rubber-band line perpendicular to the inclined plane helps you select the center of the new viewport.

4) Section

Create a drafting sectional view of solids, complete with cross-hatching. When you use SOLDRAW on a sectional view created with this option, it creates a temporary copy of the solids and uses SLICE to perform the operation

at the cutting plane that you define. SOLDRAW then generates a profile of the visible half of the solids and discards the original copy. Finally SOLDRAW sections the solids. Solids not crossing the cutting plane are generated as full profiles. Because drafting standards recommend not drawing hidden lines in sectional views, SOLVIEW freezes the View Name-HID layer.



In the original viewport, specify two points to define the sectioning plane. Define the viewing side by specifying a point on one side of the cutting plane.

Solids Draw (Soldraw)

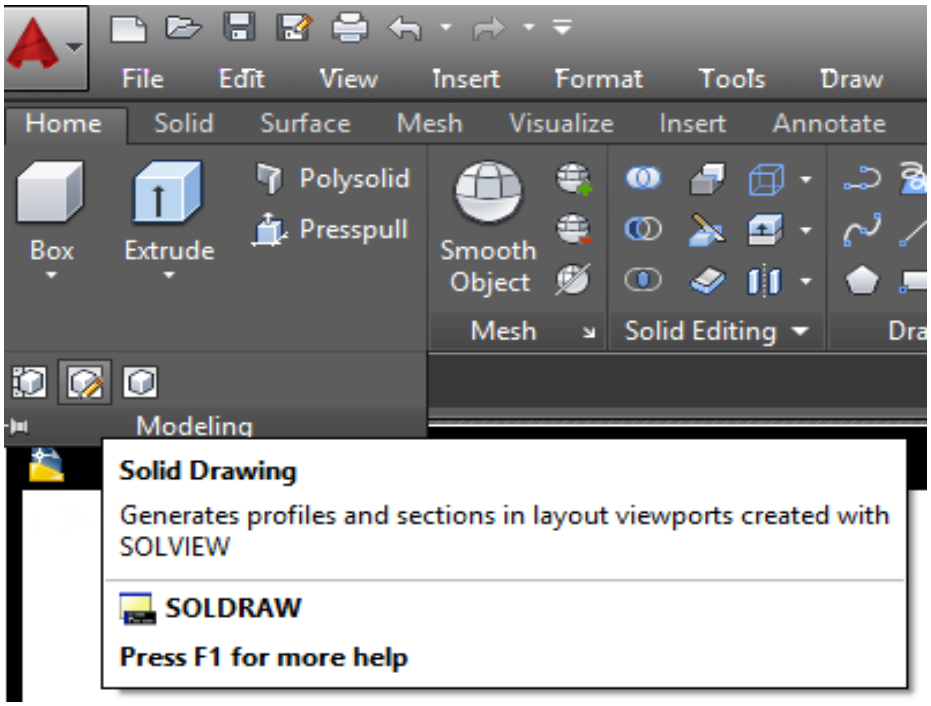
Remember

SOLDRAW is defined by the acsolids.arx application and is intended to be used only interactively. For more information, see Externally Defined Commands.

After using SOLVIEW, visible and hidden lines representing the silhouette and edges of solids in the viewport are created and then projected to a plane perpendicular to the viewing direction

Solid Draw command can be find out through both methods.

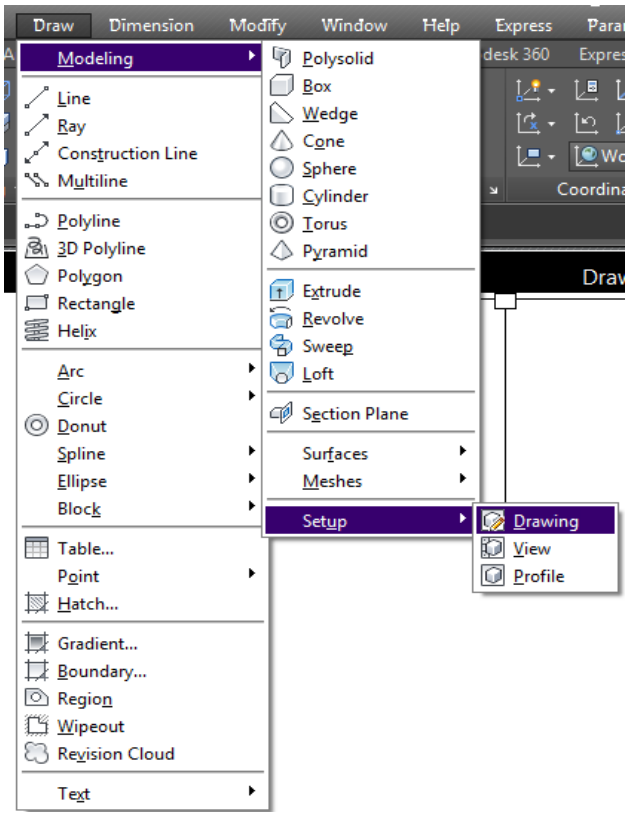
Through Ribbon



Point to be Noted

Do not place permanent drawing information on the view name-VIS, view name-HID, and view name-HAT layers. The information stored on these layers is deleted and updated when SOLDRAW is run.

Through Menu Bar



Generate profiles and sections in viewports created with SOLVIEW. SOLDRAW can only prepare viewports that have been created with SOLVIEW.

Command: Solid Draw (Shortcut key =**SOLDRAW)**

Following are the steps for this command

Step 1: Type SOLDRAW at the command prompt

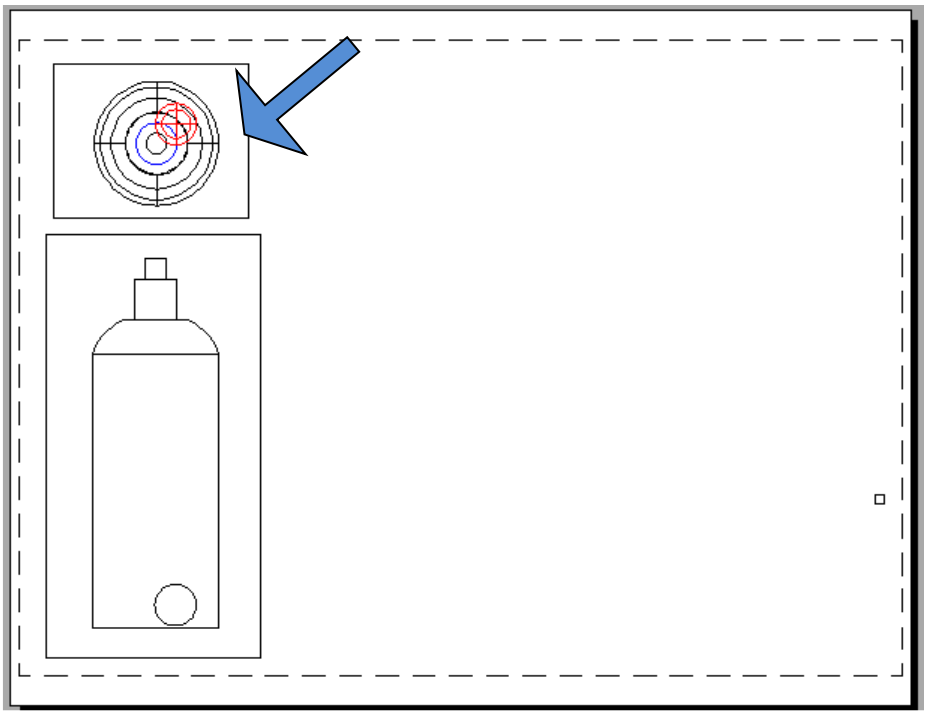
Step 2: Select viewports to draw: **Pick the front viewport created above**

Select objects: **1 found**

Step 3: Select objects: enter (3 solids selected.)

Remember

If your object was an inserted block, you need to first explode it so it becomes a solid



Option of Solid Draw

1. **Select viewports to draw**
2. **Select objects**

1. Select viewports to draw

Specifies a layout viewport that has been created using SOLVIEW. Visible and hidden lines representing the silhouette and edges of solids in the viewport are created and then projected to a plane perpendicular to the viewing direction. Silhouettes and edges are generated for all solids and portions of solids behind the cutting plane. For sectional views, cross-hatching is created using the current values of the HPNAME, HPSCALE, and HPANG system variables.

2. Select objects

Select the viewports to be drawn. Any existing profiles and sections in the selected viewport are deleted, and new ones are generated. All layers, except those required to display the profile or section, are frozen in each viewport.

Create 3D View Using UCS

Step 1: Type MSPACE and the plan view of the 3D Bottle.

Step 2: Choose View, 3D Viewport, and SW Isometric...

Step 3: Type UCS at the command prompt.

Command: **UCS**

Origin/ZAxis/3point/Object/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **VIEW**

This sets the current UCS parallel to the screen. We now need to save this **UCS**.

Step 4: Type UCS at the command prompt.

Command: **UCS**

Origin/ZAxis/3point/Object/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **SAVE**

?/Desired UCS name: **3DVIEW**

Step 5: Type PSPACE at the command prompt.

Command: **PSPACE**

Step 6: Type SOLVIEW at the command prompt.

Command: **SOLVIEW**

Ucs/Ortho/Auxiliary/Section/<exit>: **U**

Named/World/? /<Current>: **N**

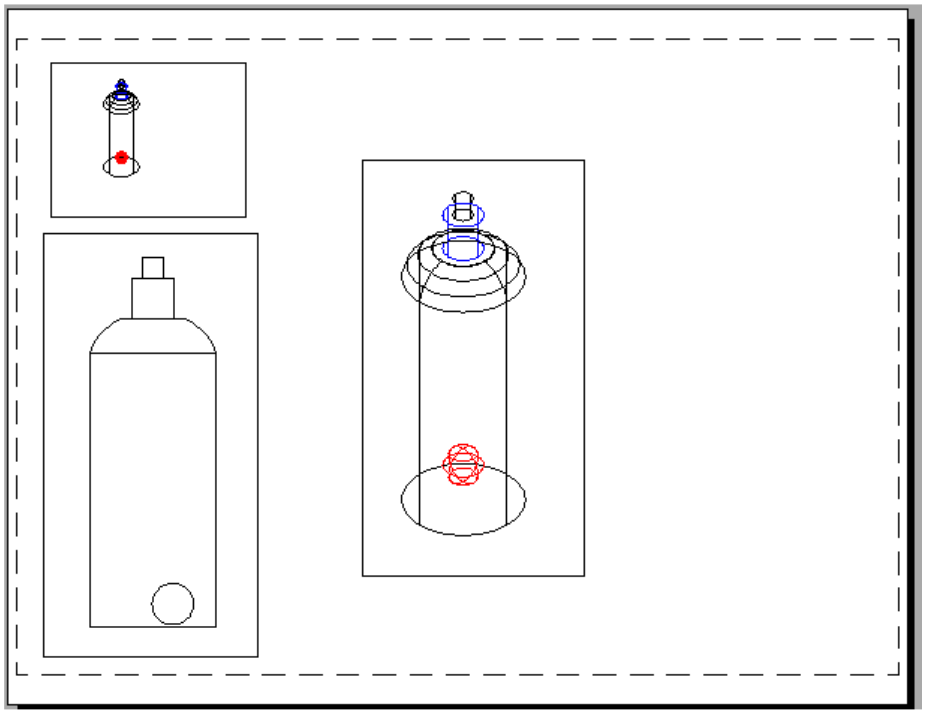
Name of UCS to restore: **3DVIEW**

Enter view scale<1.0000>: **enter**

View center: **pick upper left corner**

View center: **pick lower right corner**

Enter view name: **3DVIEW**

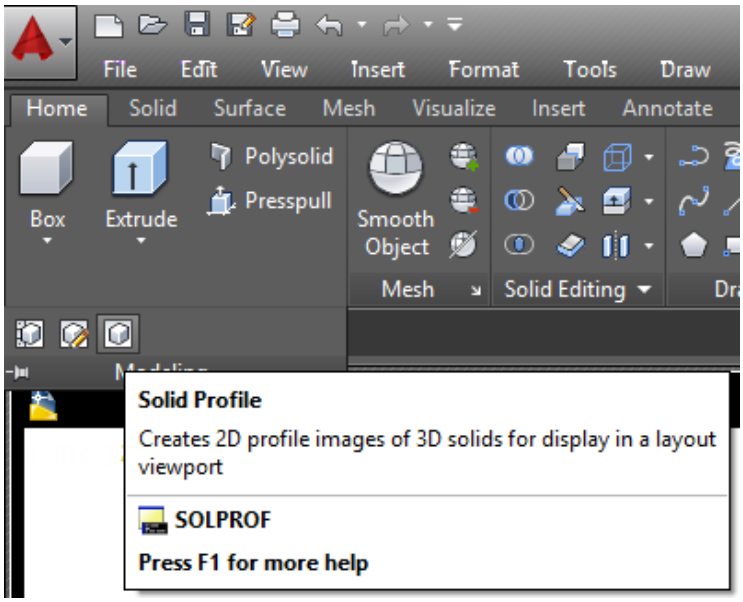


Solid Profile (Solprof)

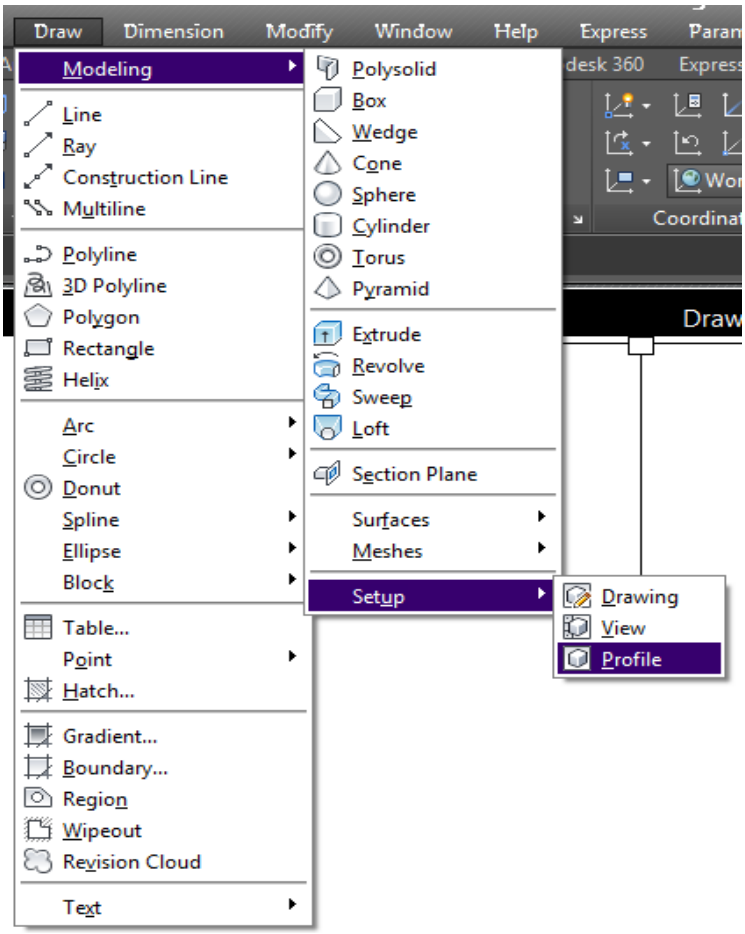
The selected 3D solids are projected onto a 2D plane parallel with the current layout viewport. The resulting 2D objects are generated on separate layers for hidden and visible lines and are displayed only in that viewport.

Solid Profile command can be find out through both ways

Through Ribbon



Through Menu Bar



Command: Solid Profile (Shortcut key **Solprof)**

Following are the steps for this command

Step 1: Double-click MODEL to return to Model Space.

Step 2: Type SOLPROF at the command prompt.

Command: **SOLPROF**

Select objects: **pick solids from the 3DView**

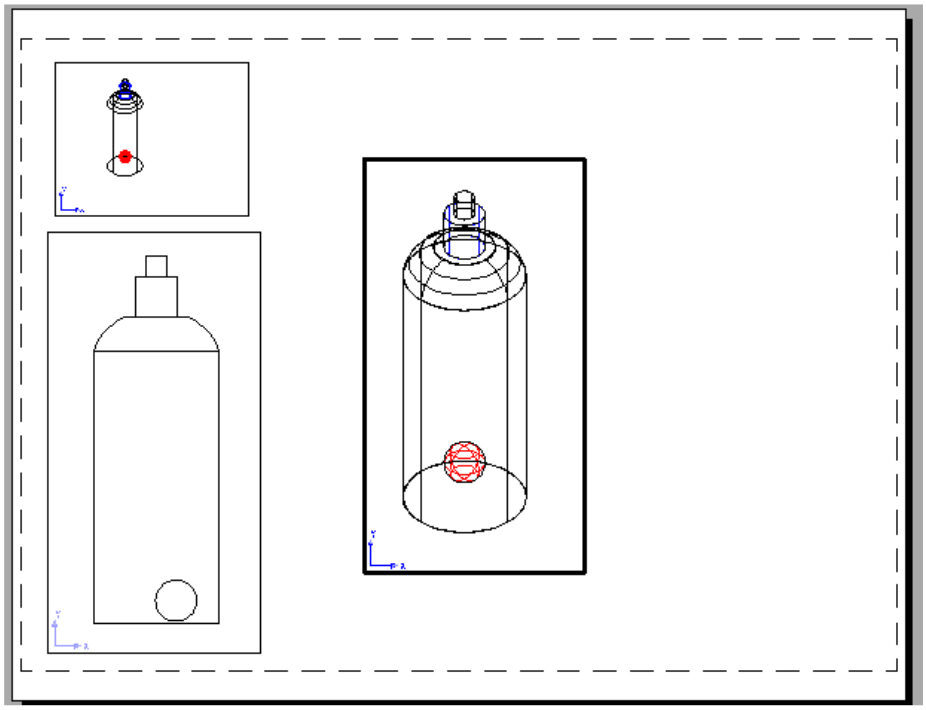
Step 3: Select objects: **enter**

Step 4: Display hidden profile lines on separate layer? <Y>: **enter**

Step 5: Project profile lines onto a plane? <Y>: **enter**

Step 6: Delete tangential edges? <Y>: **enter**

3 solids selected.



Option of Solid Profile

1. Select objects
2. Display hidden profile lines on separate layer
3. Project profile lines onto a plane
4. Delete tangential edges

1. Select Objects

Specifies the objects to be used as a basis for the profiles.

2. Display hidden profile lines on separate layer?

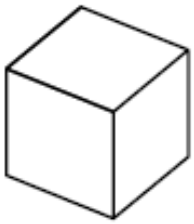
IF YES

Generate only two blocks: one for the visible lines and one for the hidden lines of the entire selection set. When you generate hidden lines, solids can partially or completely hide other solids. The visible profile block is drawn in the BYLAYER line type, and the hidden profile block is drawn in the HIDDEN line type (if loaded). The visible and hidden profile blocks are placed on uniquely named layers using the following naming conventions:

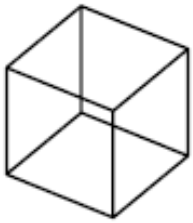
- PV-viewport handles for the visible profile layer
- PH-viewport handles for the hidden profile layer

Example

If you create a profile in a viewport whose handle is 4B, the blocks containing the visible lines are inserted on layer PV-4B, and the block containing the hidden lines (if requested) is inserted on layer PH-4B. If these layers do not exist, the command creates them. If the layers do exist, the blocks are added to the information already on the layers.



profile with hidden lines removed



profile with hidden lines displayed

IF NO

Treats all profile lines as visible lines and creates a block for the profile lines of each selected solid. All profile lines for each solid in the selection set are generated, even if a solid is partially or completely obscured by another solid. The visible profile blocks are drawn in the same line type as the original solid and placed on a uniquely named layer using the naming convention described under the Yes option.

Remember

To determine the handle of a viewport, select the viewport while in paper space and use the LIST command. Choose a layout tab to move from model space to paper space.

SOLPROF does not change the display of layers; if you want to view only the profile lines that you have created, turn off the layer containing the original solid (usually the current layer).

Remember

Solids that overlap each other (share some common volume) produce dangling edges if you request hidden-line removal. This happens because the edges must be broken at the point where they enter another solid to separate them into visible and hidden portions. You can eliminate dangling edges by combining the overlapping solids (using UNION) before generating a profile.

3. Project profile lines onto a plane?

This determines whether 2D or 3D objects are used to represent the visible and hidden lines of the profile.

IF YES

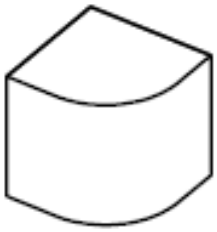
Create the profile a line with 2D objects. The 3D profile is projected onto a plane normal to the viewing direction and passing through the origin of the UCS. SOLPROF cleans up the 2D profile by eliminating lines that are parallel to the viewing direction and by converting arcs and circles that are viewed on edge into lines.

IF NO

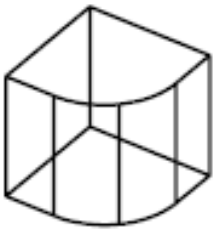
Create the profile lines with 3D objects. These tools are meant to be used in the order in which they appear in the Modeling panel and you need to follow the proper sequence of steps in order to get the desired results.

4. Delete tangential edges?

Determines whether tangential edges are displayed. A tangential edge is the transition line between two tangent faces. It's the imaginary edge at which two faces meet and are tangent. For example, if you fillet the edge of a box, tangential edges are created where the cylindrical face of the fillet blends into the planar faces of the box. Tangential edges are not shown for most drafting applications.



profile with tangential edges deleted



profile with tangential edges retained

IF YES

Delete tangential edges in the profile.

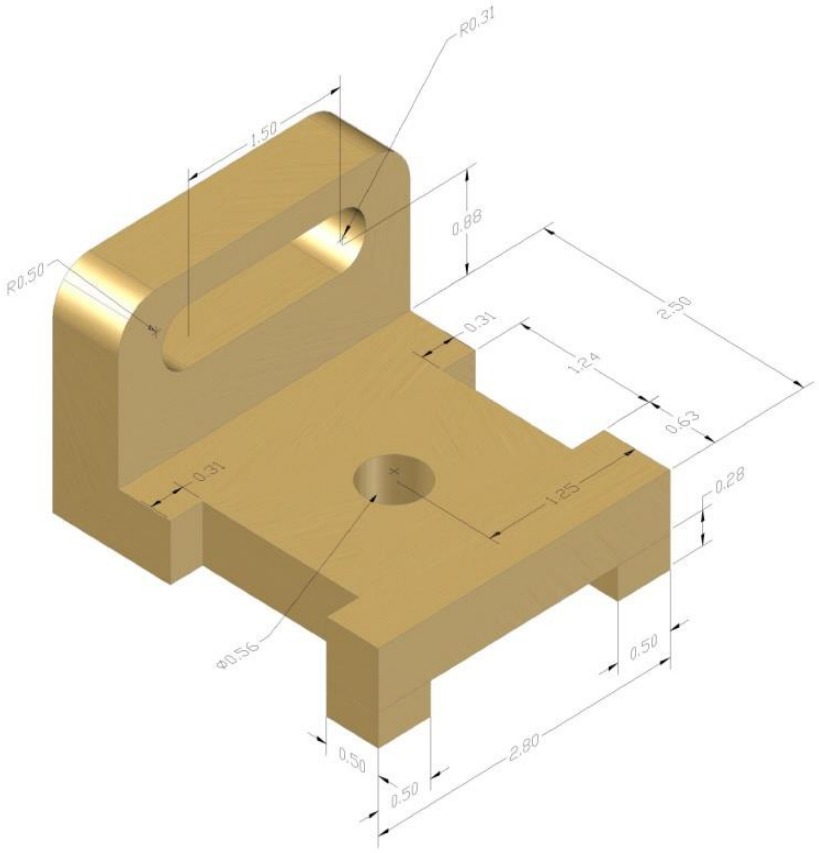
IF NO

Retain tangential edges in the profile.

Learner's Activity

1) Prepare the flattend view of the housing plane of previous Module.

2) Create a Flatshot view of the given 3D Model



3) Use Solid view, Solid draw and solid profile to create 2D view of the above given Solid.

Summary of Module

- This module has described about navigates section objects and how to create 2D geometry of 3D objects. You have learnt about the generation and manipulation of 2D and 3D sections. You have learnt how to define section planes based on 3D module and about the use of section planes in order to view internal section of 3D modules, creation of 2D cross section of modules. You also have learnt about the generation of 2D geometry containing mass property information and use of this mass property information in strength and area calculations.

- You have learnt about the generation of flattened view of 3D models and about the flat short dialogue in order to create 2D geometry of your 3D models as this flat shot dialogue box helped you to control the settings of your 2D geometry. You have learnt how to turn off or frozen the objects that you do not want to capture in flat shot and about the creation of tangential lines, naming of your 2D view and isometric view. You also have learnt about the use of solid view, solid draw and solid profile tools to create 2D geometry of solids.

Frequently Asked Questions (FAQs)

FAQ 1: What is the Section?

Answer

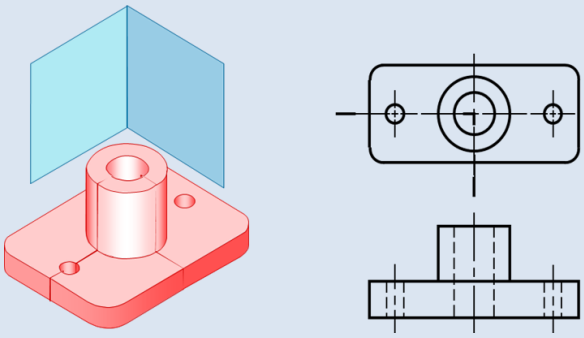
Section objects have a transparent section plane indicator that acts as a cutting plane. Because it is transparent, it's a useful visual tool as it lets you to see the geometry on either side of the cutting plane. This plane can be easily placed and moved anywhere throughout a 3D model that is composed of 3D solids, surfaces, or regions (two-dimensional areas created from closed shapes or loops

FAQ 2: Differentiate between Half Section and Full Section?

Answer

Half section

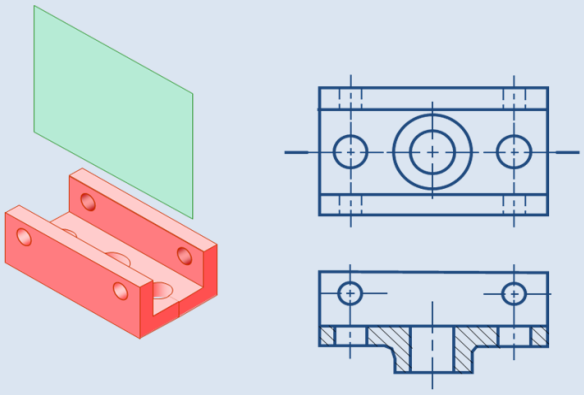
The view is made by passing the cutting plane halfway through an object and removes a quarter of it.



A center line is used to separate the sectioned half from the unsanctioned half of the view. Hidden line is omitted in unsection half of the view.

Full section

The view is made by passing the straight cutting plane completely through the part.



FAQ 3:	What are the Section plan Options?
Answer	Options of Section plane 1. Face or Any Point to Locate Section Line 2. Draw section 3. Orthographic
FAQ 4:	What is the Section Plane?
Answer	Section plane objects create sections of 3D solids, surfaces, meshes, and point clouds. Use live sectioning with section plane objects to analyze a model, save sections as blocks for use in layouts, and extract 2D geometry from point clouds.
FAQ 5:	What is the different between point on desired side & keep Both sides in Slice Command
Answer	• Point on desired side This point is use to determine which side of the sliced solids, your drawing retains. This point cannot lie on the cutting plane. • Keep both sides Retain both sides of the sliced solids. Slicing a single solid into two pieces creates two solids from the pieces on either side of the plane
FAQ 6:	How many option are there for Slice command Option
Answer	It's are eight option 1) Planar object 2) Surface 3) Z-axis 4) View 5) XY 6) YZ 7) ZX 8) 3points

FAQ 7:	What is a Flattened View?
Answer	A flattened view can be defined as a view created from a 3D model in which objects are projected in their current position, relative to the view, onto a plane that is parallel to the current view. Flattened views can be from any angle and are not limited to standard orthographic projections. In most cases, it is equally beneficial to include a flattened view of an isometric projection. The following illustration shows how a 3D model is projected onto a viewing plane, resulting in a 2D Elevation
FAQ 8:	What do you know about Flatshot Command?
Answer	Flatshot tool (FLATSHOT command) is used to create a two-dimensional representation of a 3D object. When you use this tool, AutoCAD projects the edges of all 3D solids, surfaces, and meshes in the drawing onto a plane parallel to the current viewing plane. The 2D representation of these edges can then be inserted as a block on the XY plane of the current UCS or saved as a separate drawing.
FAQ 9:	What is the Solid View Command?
Answer	This command automates the Manual process of creating views, layers, and layout viewports for 3D models.
FAQ 10:	How many ways are there to select the Solid profile command?
Answer	Solid Profile command can be find out through both ways • Through Ribbon • Through Menu Bar

Test Yourself!

Please mark the correct one from the given options. You can check your answer with the Answer Key at the end of this module.

1. How many types of Section?

a. Four
b. Six
c. Five
d. Seven

2. How many sections Plane Status?

a. Three
b. Four
c. Two
d. Five

3. Short cut key for Slice Command is….

a. S
b. Sli
c. Sl
d. None Of these

4. Short cut key for Solid Draw Command is….

a. Solddraw
b. Sodraw
c. Soldraw
d. Sdraw

5. What is the shortcut key for solid profile?

a. Solprof
b. Soliprof
c. Sprof
d. None of these

Answer Key

MCQ Number	Correct Answer
1	b
2	a
3	c
4	c
5	a

AUTOCAD

Module-11

LEARNING GUIDE

National Vocational Certificate Level 3

Version 1 - July 2013

Module 11: Rendering, Materials and Lights

Learning Outcomes

After completion of this learning module, you will be able to:

- Execute Rendering
- Apply/Configure Materials
- Apply Lights

Learning Unit 1: Execute Rendering

Overview

This unit describes familiarity with AutoCAD Rendering. After completion of this learning unit, the learner will be able to work with different interface like interior and exterior with 3D modeling along with its Ribbons and Pull down menu. Render command, destination, quality, selection, crop, and file are the main functions by using **RPREF** command will discuss in this unit.

Further you will be to understand the Advance rendering setting and execute the process to render to file and turn on/off render to file and will be able to apply environmental features to set up atmospheric effects or background images using “RENDERENVIRONMENT” command and different backgrounds by generating new view.

1. Introduction to Render

Render is used to get the most realistic and photorealistic view of your 3D drawing by adding realistic materials.

2. Create a Photorealistic or Realistically

Here we are going to view the default render window without applying material and more rendering setting.

You can get render by using the following methods

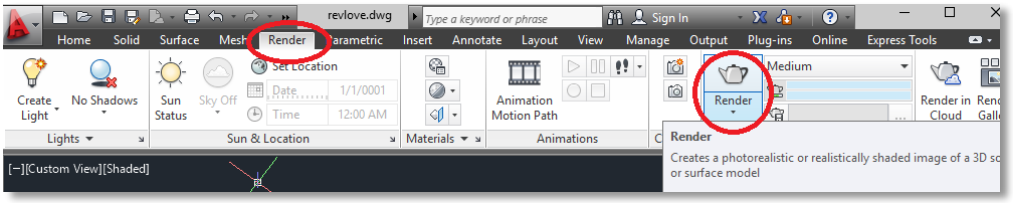
Method-1:

Type **render** command in command bar

Method-2:

Remember

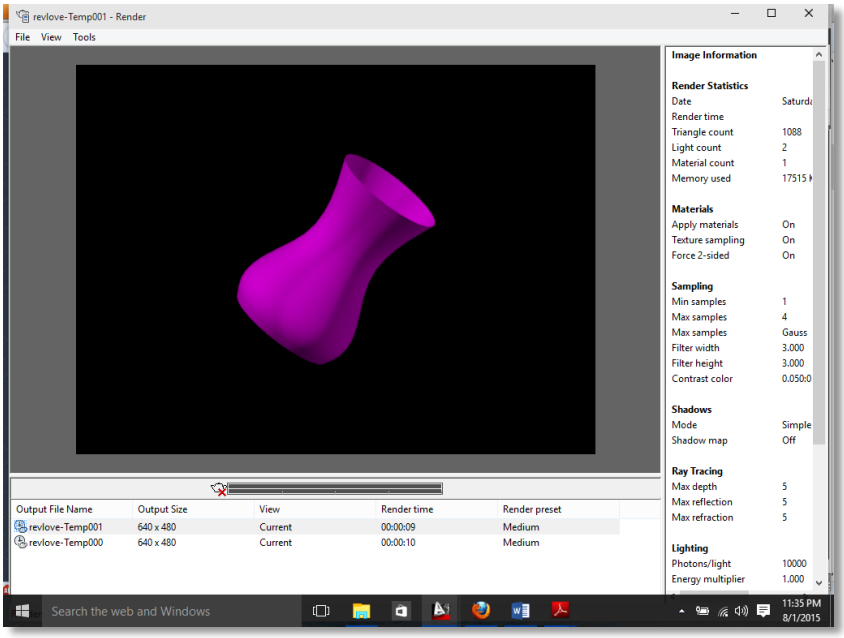
If the Render Option is not displayed on the Visualize tab, right-click the Visualize tab and click Panels →Render.



Result of default Rendering

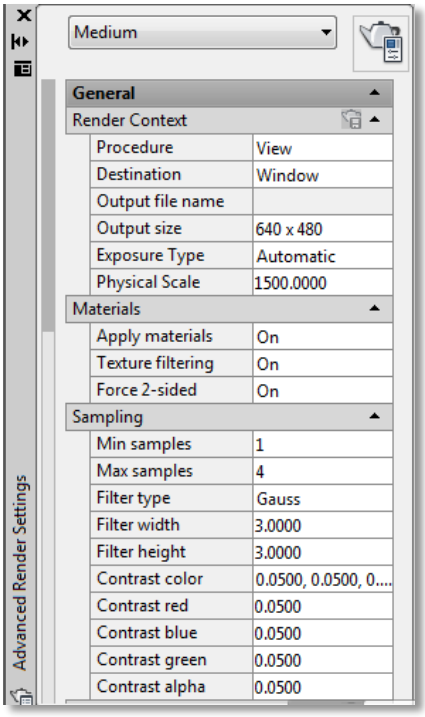
Remember

Advance Render Settings palette contains all the primary control for the rendering.



3. Using “RPERF” Command”

RPERF command is used to hide or unhide the Advanced Render Settings palette for access to advanced rendering settings.



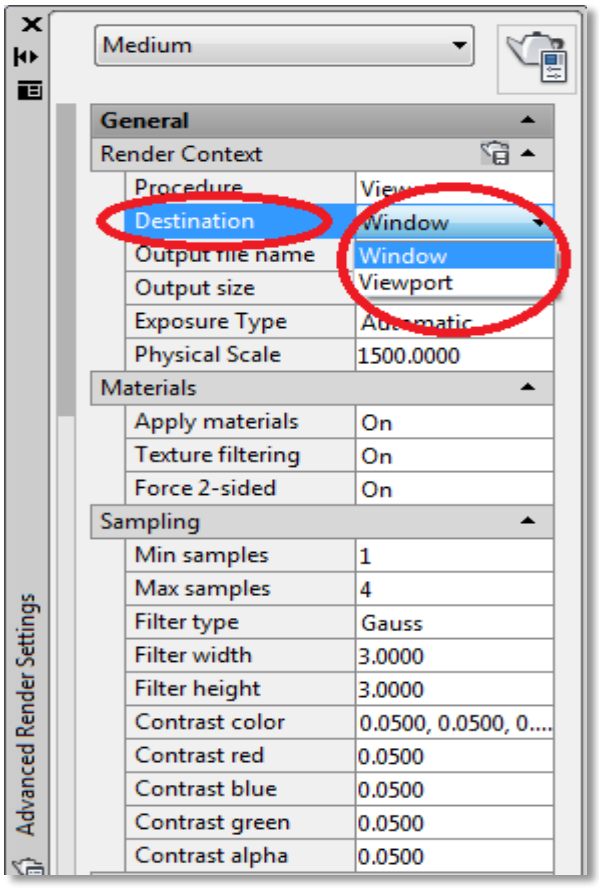
4. Destination

Here we are going to make some modification in Advance Render Settings palette for the selection of **"DESTINATION".**

Destination defines the output site that the renderer uses to display the rendered image.

There are two sites

- **Window**. Renders to the Render Window.
- **Viewport**. Renders to a viewport.

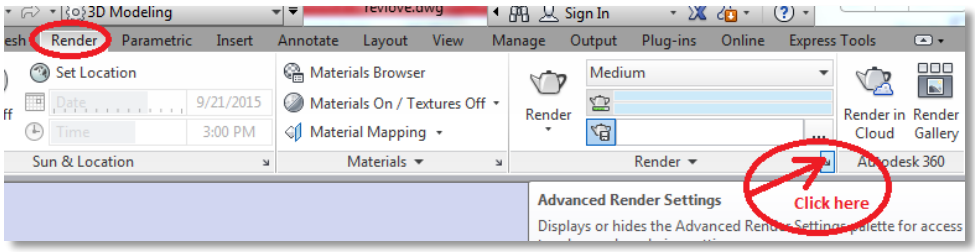


To access

Advance Render Settings by the following methods

Method-1: Type **RPERF** in command bar.

Method-2:



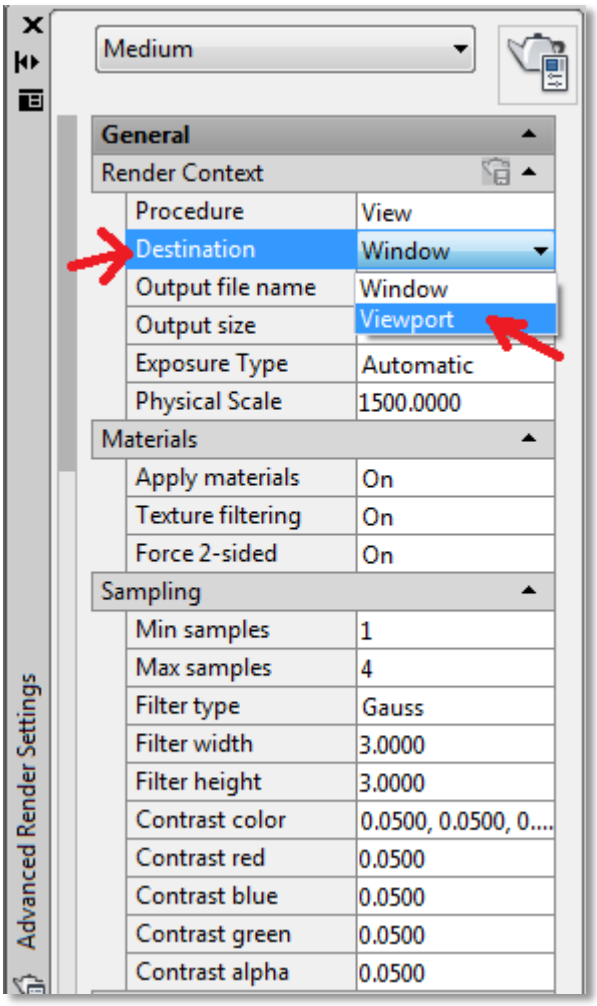
Procedure
Apply Advanced Render Settings

STEP BY STEP Operation on AutoCAD

GET READY. Before you begin these steps, be sure to set unit and you must have an objects in your drawing to apply rendering.

Step-1: Type **RPERF** command in command bar <Enter>

Step-2: Select destination field and switch into viewport as shown below



Step-3: Type **render** command in command bar.

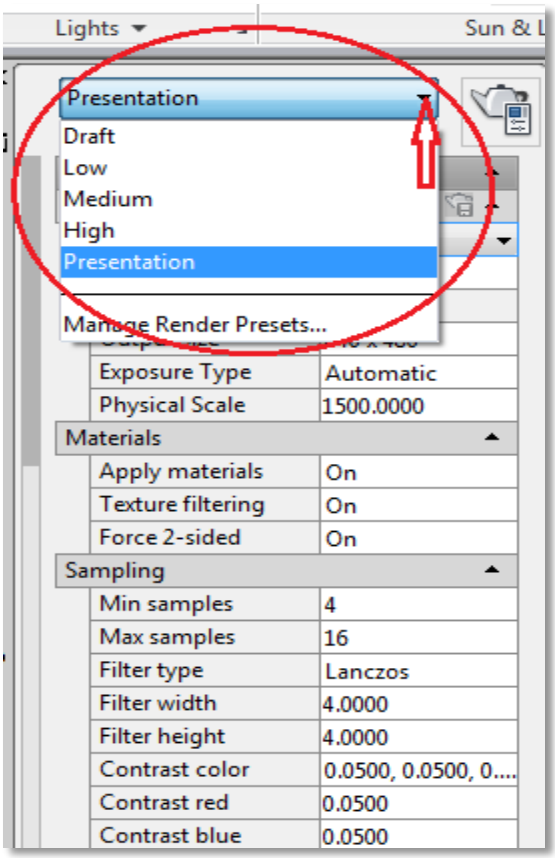
Step-4: Repeat the above steps for Window destination.

Deference between window and viewport: A separate window is open for window destination which you can save for future use while viewport destination rendered the drawing in the same window, which cannot be save and for temporary display.

5. Quality Setting

You can set the quality of image for rendering by using Advance Render Setting as shown in figure below

Note: Quality of the rendering to be open takes time according to material, lighting etc.



6. Render Procedure

Render procedure Control the parts of the model that gets processed during rendering for following three settings

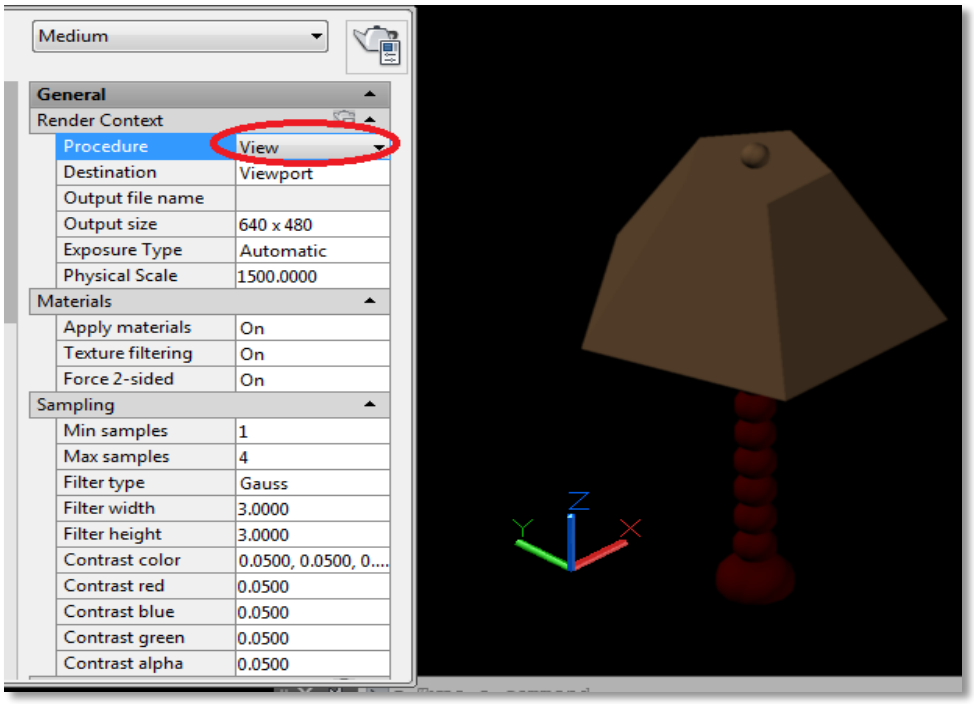
- **View**. Allow you to renders the current view without displaying the Render dialog box.
    - **Crop**. Allow you to create a render area at render time to specify area to render before rendering proceeds. This option is available only when Viewport is selected under Destination.
    - **Selected**. Allow you to displays a prompt to select objects to render.

View Procedure

Step-1: Type **RPERF** command in command bar <Enter>

Step-2: Select Procedure field and switch into view as shown below

Step-3: Apply render by using **render** command

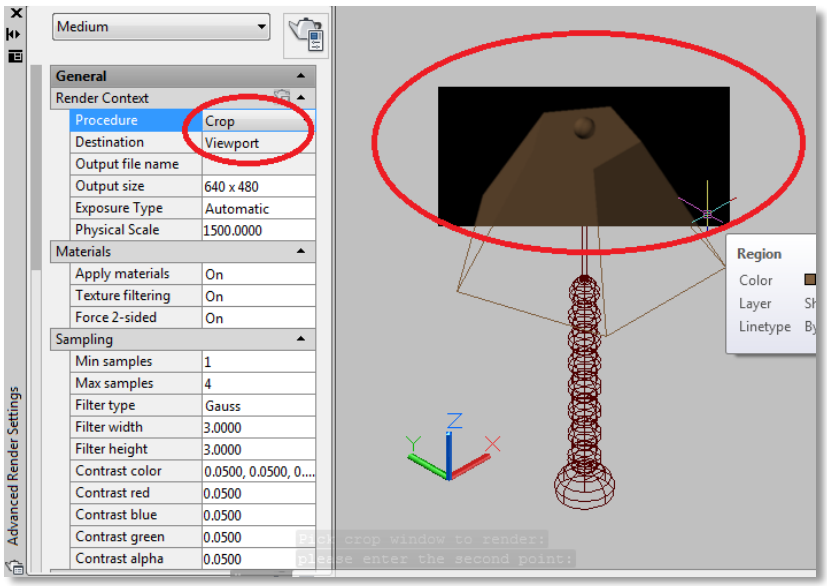


Crop Procedure

Step-1: Type **RPERF** command in command bar <Enter>

Step-2: Select Procedure field and switch into crop as shown below

Step-3: Apply render by using **render** command

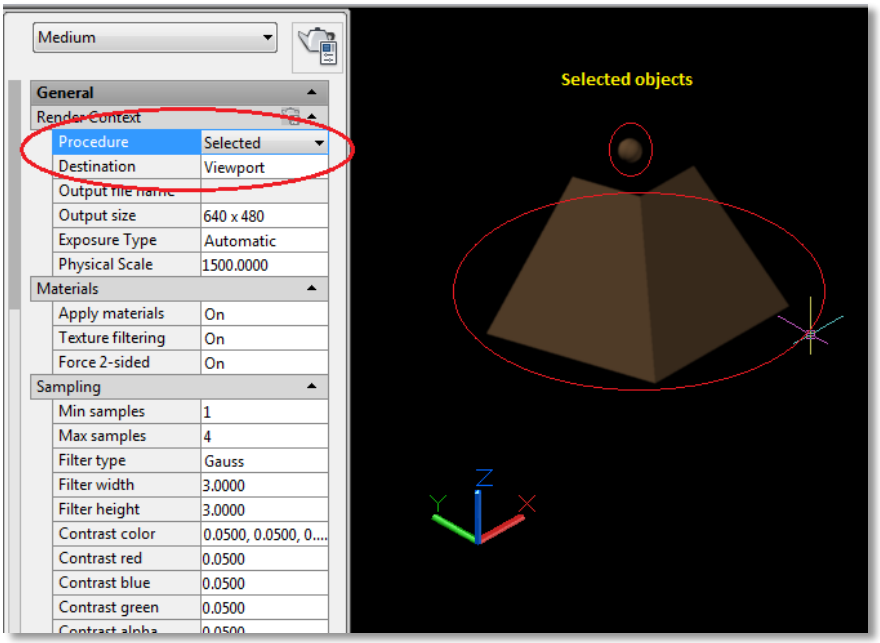


Selected Procedure

Step-1: Type **RPERF** command in command bar <Enter>.

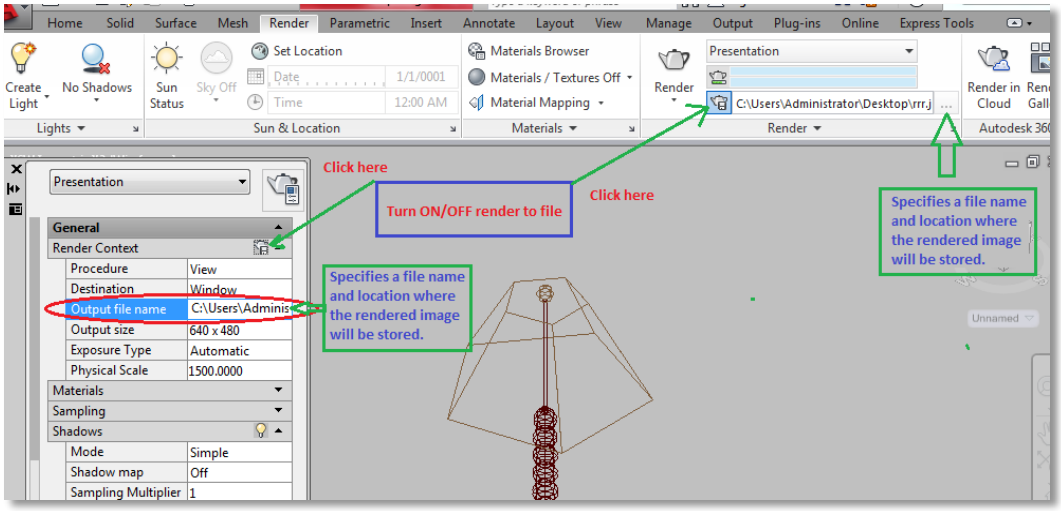
Step-2: Select Procedure field and switch into Selected as shown below.

Step-3: Apply render by using **render** command.



Render to File and Turn off Render to File.

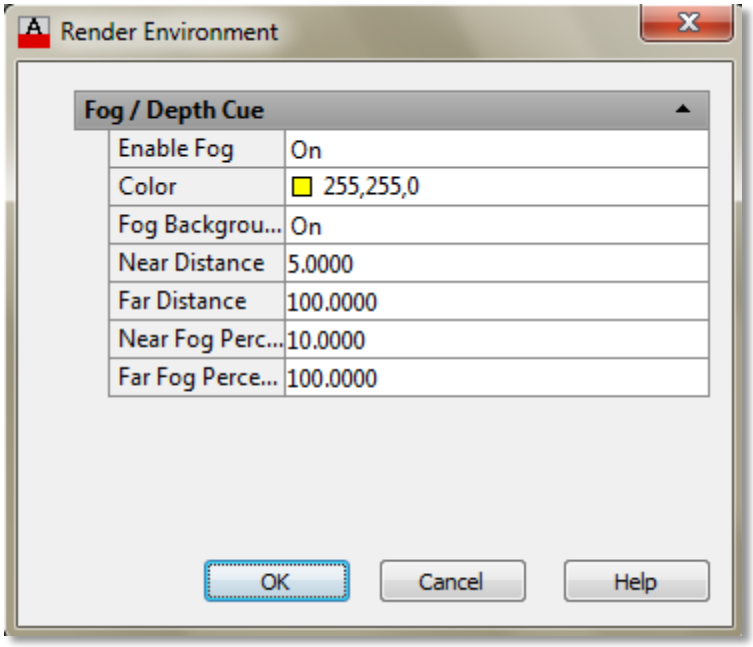
You can turn **on/off Render** to file by the following procedure.



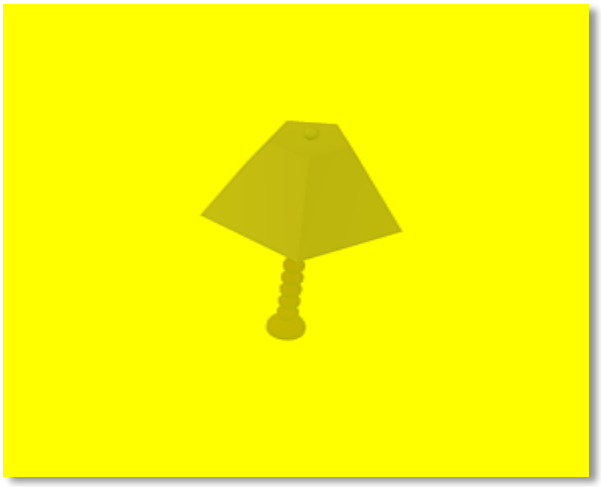
RENDERENVIRONMENT

Create **fog** effect to enhance sense of distance.

Step-1: Type **RENDERENVIRONMENT** command in command bar <Enter>



Step-2: Apply **render**

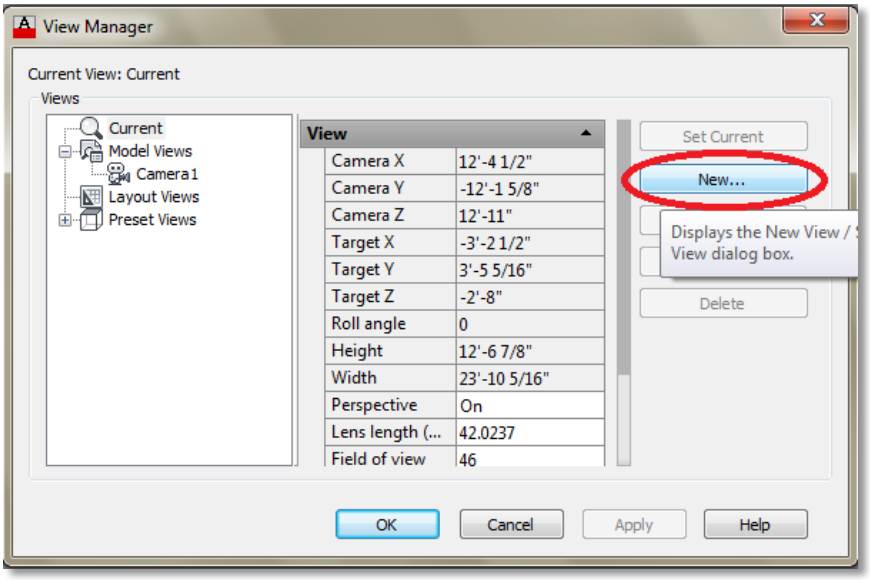


Creating Views and Applying Background

View is used to create, set, rename, modify, and delete named view, including model named views and camera views, layout views, and preset view.

Here we are using view command or view manager to set different background.

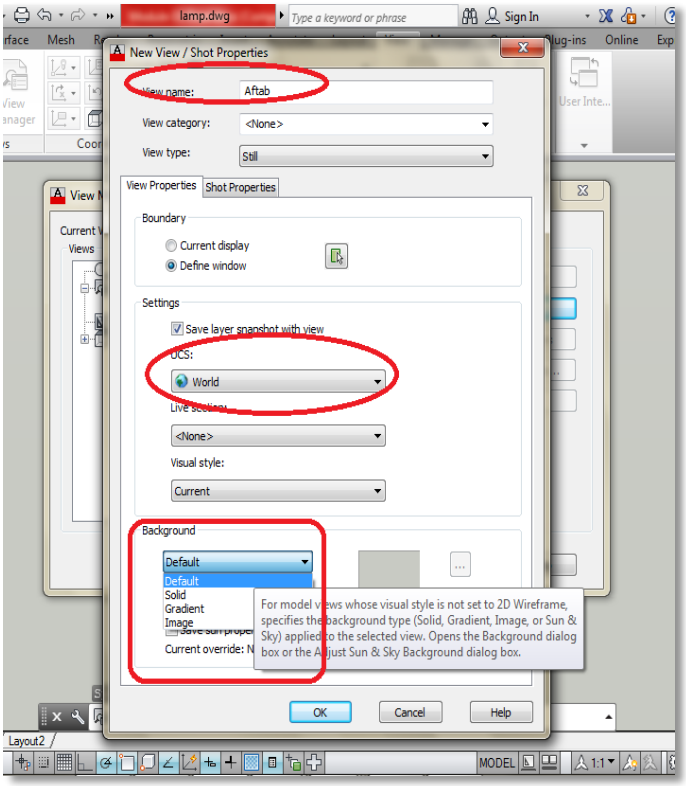
Step-1: Type **view** command in command bar or click View tab→views Manager to get the following.



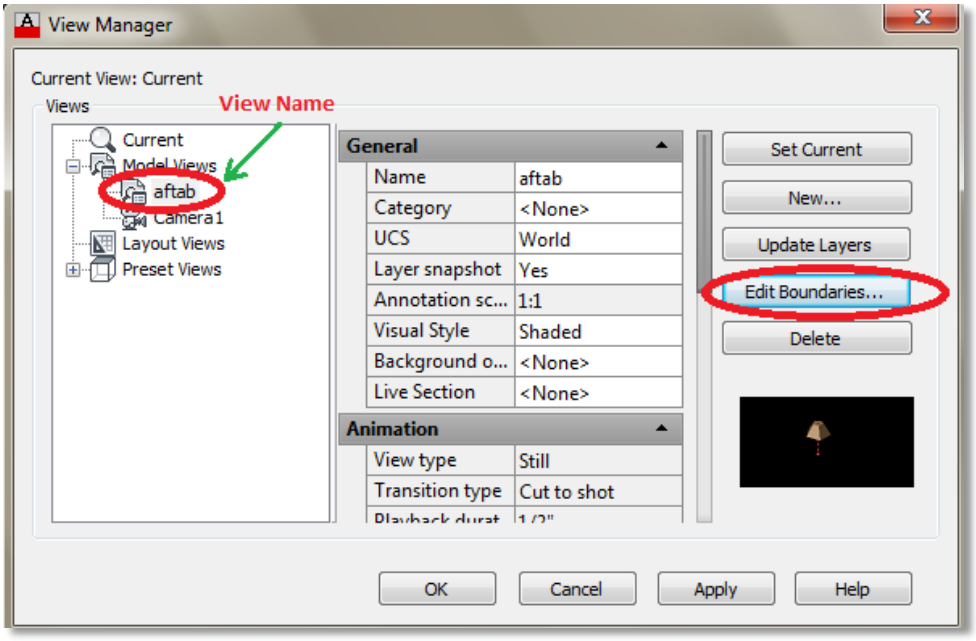
Step-2: Click the new button and get the following properties setting as shown in the figure below.

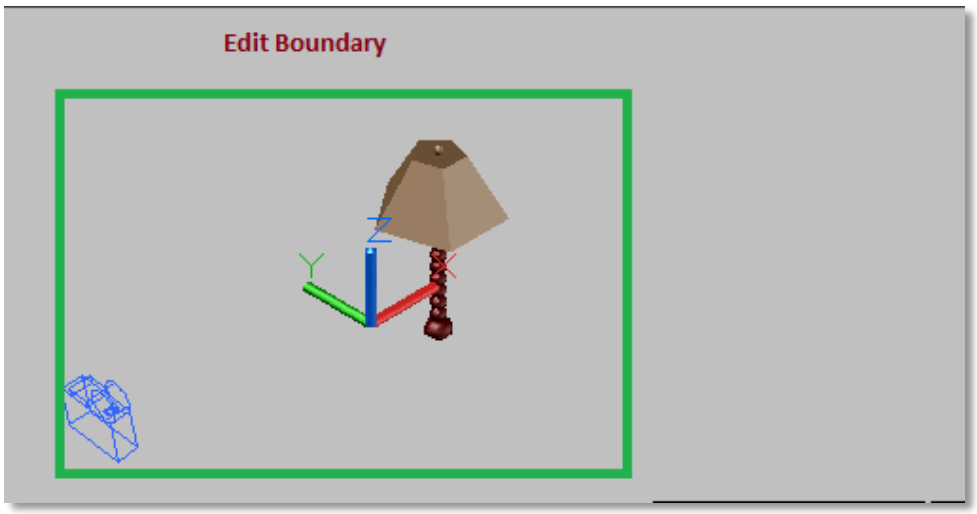
Note:
You can change the setting according to your View.

Here we applied:
View Name: Aftab
UCS: World

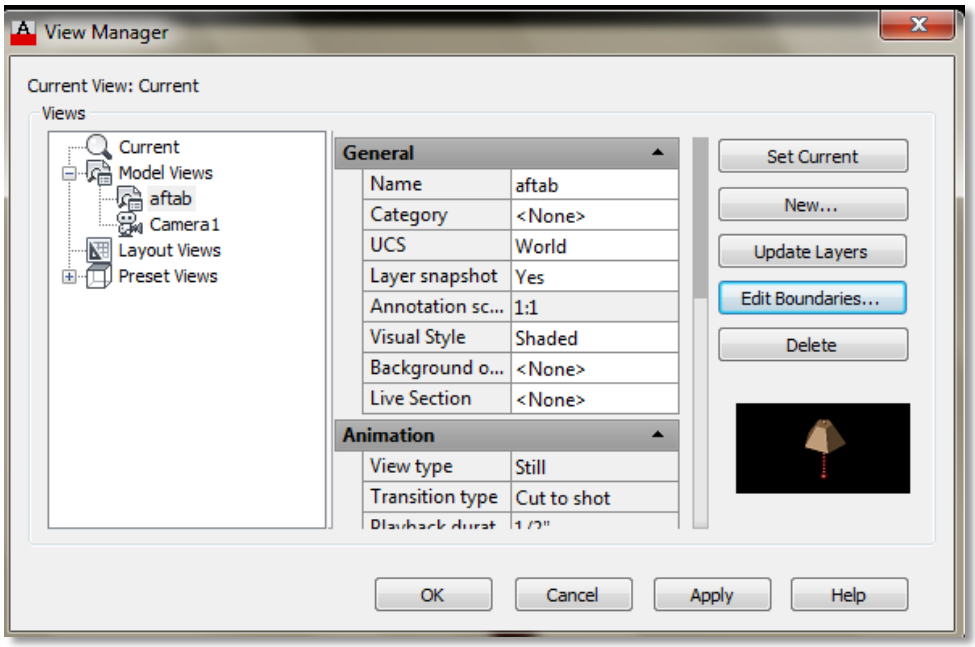


Step-3: Click the Edit Boundary option to define your own boundary for the background as shown in the figure below.

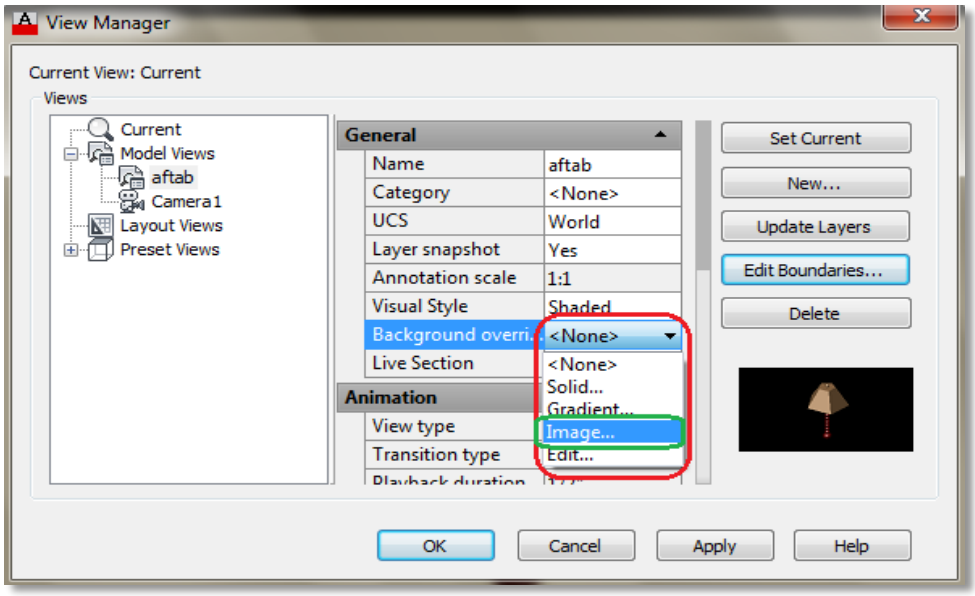




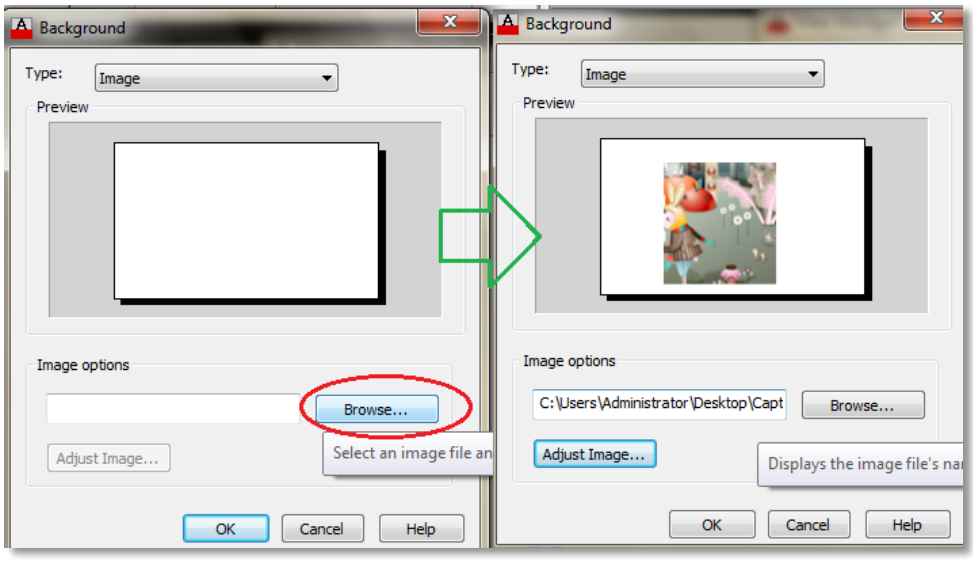
Step-4: After defining the boundary press Enter, you will get back to View Manager Window.



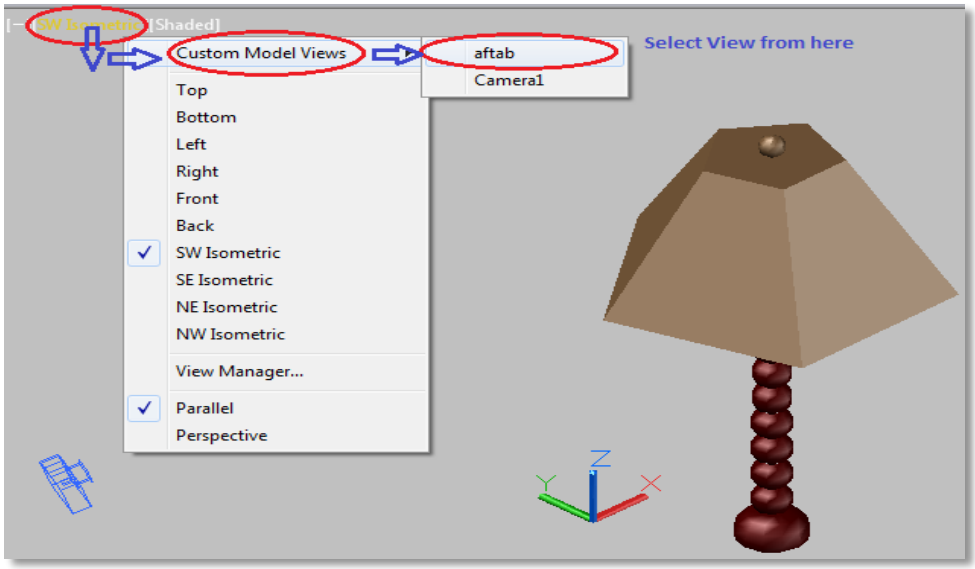
Step-5: Now select background option to apply one of the three option of background as show in figure below.



Step-6: Here we are going select the image option to select image from file and click **Ok** as shown in figure.



Step-7: Select view from view control as follow



You should get the follow Out Put

Type a command

Learner's Activity

Apply Solid background with single and multicolor to your drawing.

Note:
Materials with its properties can be apply c

Materials

Material contains settings that affect how materials are handled by the renderer.

Apply Materials

Apply the surface materials that you define and attach to an object in the drawing. If Apply Materials is not selected, all objects in the drawing assume the color, ambient, diffuse, reflection, roughness, transparency, refraction, and bump map attribute values defined for the GLOBAL material.

Texture Filtering

Specify how texture maps are filtered.

Force 2-Sided

Control if both sides of faces are rendered.

Control Sampling

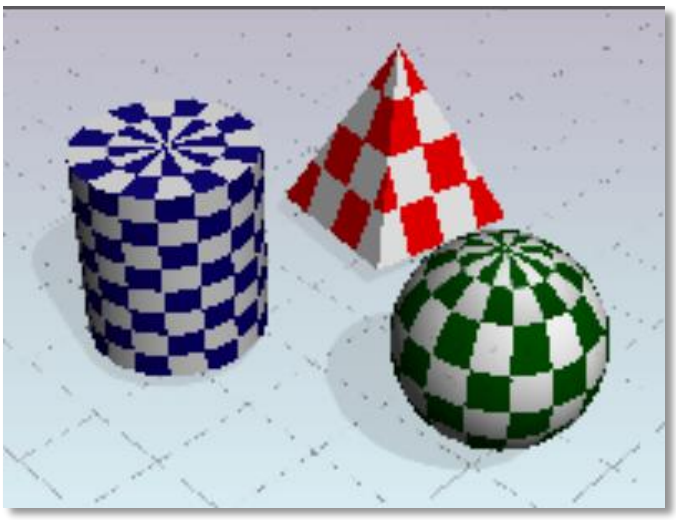
Control how the renderer performs sampling.

Sampling provides a "best guess" color for each rendered pixel. The renderer first sample the scene color at locations within the pixel or along the pixel's edge, then uses a filter to combine the samples into a single pixel color. Increasing the minimum and maximum sample ranges greatly improves the quality of a rendering.

If image quality is highly degraded

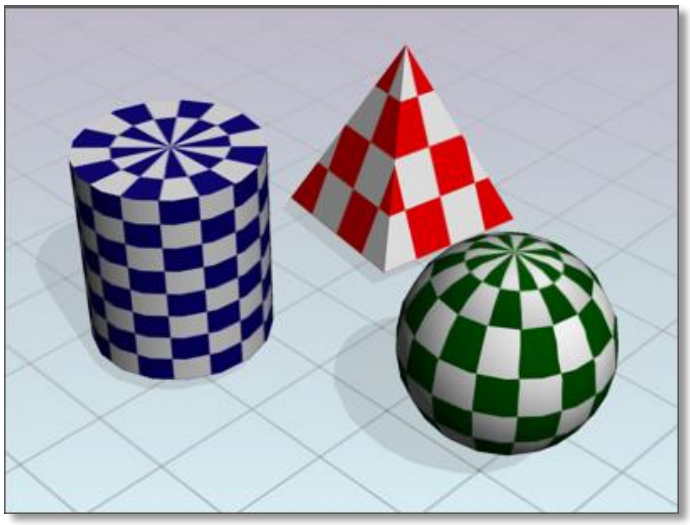
- Min. Samples = 1/64;
- Max. Samples = 1/4.

Note:
Controls sampling can be applied.



In the following example, increased sampling smooth's edges

- Min. Samples = 1;
- Max. Samples = 16.

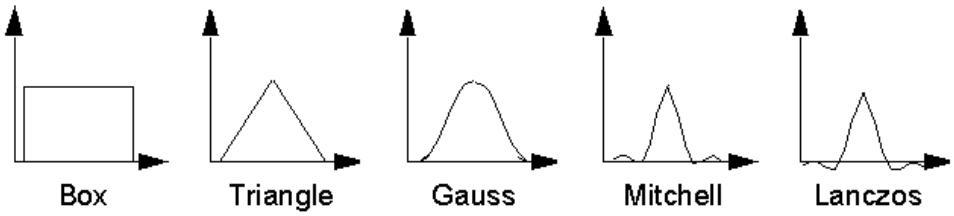


Filter Type

There are five filter methods:

- Box, the default, is the quickest sampling method.
- Mitchell is often the most accurate.

- Box
- Gauss Triangle
- Mitchell
- Lanczos.



Filter Width and Filter Height

Specify the size of the filtered area. Increasing the value of Filter Width and Filter Height can soften the image; however, it will increase rendering time.

Contrast Red, Blue, Green

Specify the threshold values for the red, blue, and green components of samples. These values are normalized, and range from 0.0 to 1.0, where 0.0 indicates the color component is fully unsaturated (black, or 0 in eight-bit encoding) and 1.0 indicates the color component is fully saturated (white, or 255 in eight-bit encoding).

Contrast Alpha

Specify the threshold value for the alpha component of samples. This value is normalized, and ranges from 0.0 (fully transparent, or 0 in eight-bit encoding) to 1.0 (fully opaque, or 255 in eight-bit encoding).

Note:
Shadows with its properties can be applied

Shadow

Contains settings that affect how shadows appear in the rendered image.

Mode

- **Simple**. Generates shadow shaders in a random order.
- **Sort**. Generates shadow shaders in order, from the object to the light.
- **Segments**. Generates shadow shaders in order along the light ray from the volume shaders to the segments of the light ray between the object and the light.

Shadow Map

Control if shadow mapping is used to render shadows. When on, the renderer renders shadow-mapped shadow. When off, all shadows are ray-traced.

Sampling Multiplier

Globally limits shadow sampling for area lights.

Ray Tracing

Ray tracing contains settings that affect the shading of a rendered image, which effects of reflection and refraction.

Max Depth

Limit the combination of reflection and refraction.

Max Reflections

Set the number of times a ray can be reflected.

Max Refractions

Set the number of times a ray can be refracted

Illumination

Do you know!

In 3D computer graphics,

RADIOSITY is an application of the finite element method to solving the rendering equation for

Scenes with surfaces that reflect light diffusely.

Indirect illumination techniques, like global illumination and final gathering, enhance the realism of a scene by simulating radiosity, or the inter-reflection of light in a scene.

Enable

Specify if lights should cast indirect light into the scene.

Photons/Samples

Set how many photons are used to compute the intensity of the global illumination. Increasing this value makes global illumination less noisy but also more blurry. Decreasing this value makes global illumination more noisy but less blurry. The larger the Samples value, the greater the rendering time.

Note:
Illumination with its properties can be applied

Determine the size of photons. When on, the spinner value set the size of photon. When off, each photon is calculated to be 1/10 of the radius of the full scene.

Radius

Specifies the area within which photons will be used when luminance is computed.

Max Depth

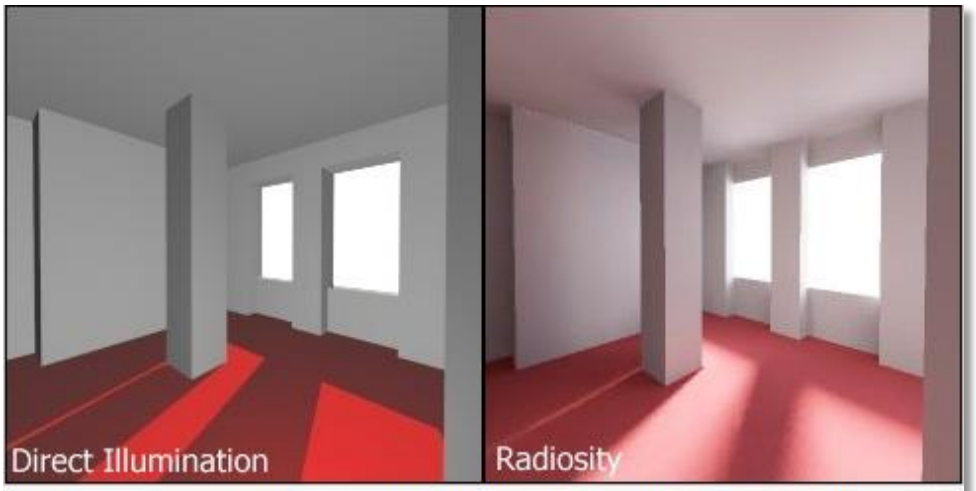
Limit the combination of reflection and refraction. Reflection and refraction of a photon stop when the total number of both equals the Max Depth setting. For example, if Max Depth equals 3 and the trace depths each equal 2, a photon can be reflected twice and refracted once, or vice versa, but it can't be reflected and refracted four times.

Max Reflections

Set the number of times a photon can be reflected. At 0, no reflection occurs. At 1, the photon can be reflected once only. At 2, the photon can be reflected twice, and so on.

Max Refractions

Set the number of times a photon can be refracted. At 0, no refraction occurs. At 1, the photon can be refracted once only. At 2, the photon can be refracted twice, and so on.



Difference between standard direct illumination without shadow umbra, and radiosity with shadow umbra

Diagnostic

Diagnostic also called visual to help you understand why the renderer is behaving in a certain way.

Grid:

Render an image that shows the coordinate space of objects, the world, or camera.

- **Object**. Shows local coordinates (UVW). Each object has its own coordinate space.
- **World**. Shows world coordinates (XYZ). The same coordinate system applies to all objects.
- **Camera**. Shows camera coordinates, which appear as a rectangular grid super imposed on the view.

Grid Size

Set the size of the grid.

Photon

Render the effect of a photon map. This requires that a photon map be present. If no photon map is present, the Photon rendering looks just like the non-diagnostic rendering of the scene: the renderer first renders the shaded scene, then replaces it with the pseudo-color image.

- **Density**. Renders the photon map as it is projected into the scene. High density is displayed in red, and lower values render in increasingly cooler colors.

- **Irradiance**. Similar to the Density rendering, but shades the photons based on their irradiance. The maximum irradiance is rendered in red, and lower values render in increasingly cooler colors.

BSP

Render a visualization of the parameters used by the tree in the BSP ray-trace acceleration method. If a message from the renderer reports excessively large depth or size values, or if rendering seems unusually slow, this can help you locate the problem.

- **Depth**. Shows the depth of the tree, with top faces in bright red, and increasingly deep faces in increasingly cool colors.
- **Size**. Shows the size of leaves in the tree, with differently sized leaves indicated by different colors.

Processing

Tile Size

Determine the tile size for rendering. To render the scene, the image is subdivided into tiles. The smaller the tile size, the more image updates are generated during rendering. When the tile size is reduced, the number of image updates increases, meaning that a rendering take longer to complete. If the tile size is increased, fewer image updates occur and the rendering takes less time to complete.

Tile Order

Specifies the method used (render order) for tiles as an image is rendered. You can choose a method based on how you prefer to see the image appear as it renders in the Render Window.

- **Hilbert**. Next tile to be rendered is based on the cost of switching to the next one.
- **Spiral**. Tiles are rendered beginning at the center of the image, and spiral outward.
- **Left to Right**. Tiles are rendered in columns, from bottom to top, left to right.
- **Right to Left**. Tiles are rendered in columns, from bottom to top, right to left.
- **Top to Bottom**. Tiles are rendered in rows, from right to left, top to bottom.
- **Bottom to Top**. Tiles are rendered in rows, from right to left, bottom to top.

Learning Unit 2: Apply/Configure Materials

Overview

This unit describes understanding of AutoCAD Materials configuration to an object in the drawing, after completing this unit you will be able to create and apply material. You will also be able to understand a material is simply an image stretched over an object to make it appear as if the object is made out of various materials such as wood, marble, brick, metal, plastic, glass, and so on. AutoCAD software provides a materials library that you can attach to the objects. Once you have applied the material, you will learn how to adjust how the material is mapped to the object. Using visual styles, you can immediately see the effects of the materials, or you can see the full effects by rendering the scene. This unit will explore the methods that help you to create new materials that you can add into your scene, photo, images etc.

1. Add Material to Drawing

A material is simply an image stretched over an object to make it appear as if the object is made out of various materials such as wood, marble, brick, metal, plastic, glass, and so on.

To Browse Material

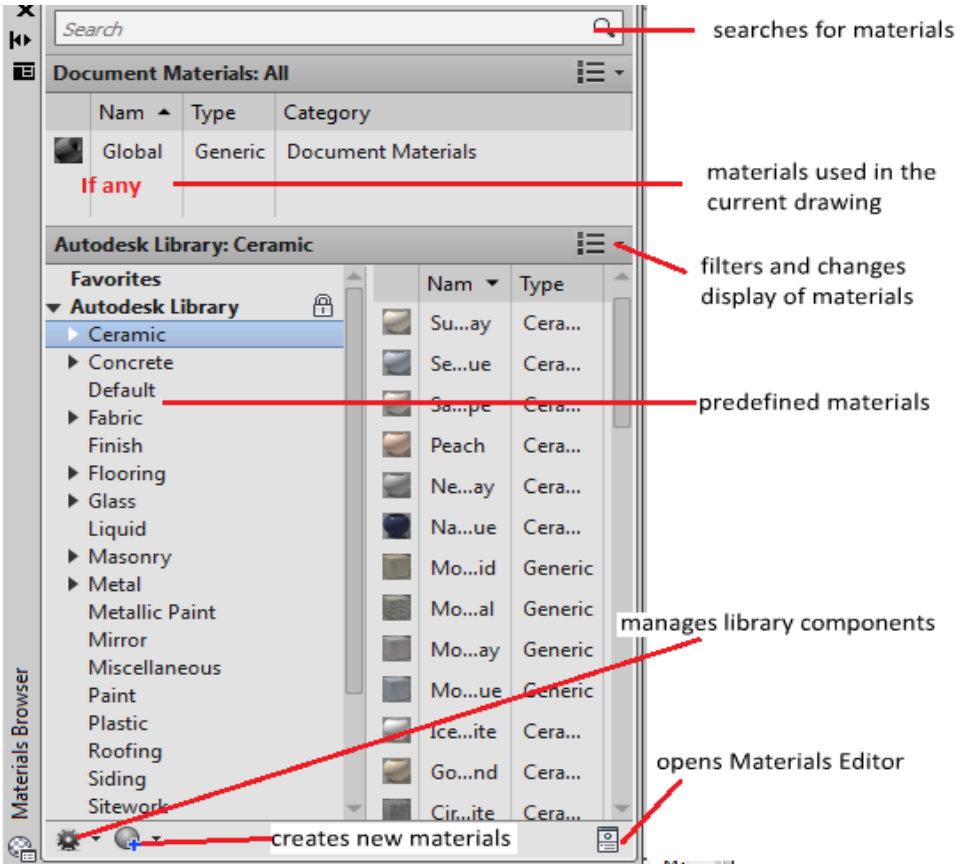
Render Tab	Button	Command Entry
Material→Browse Material.		Marbrowseropen

Procedure

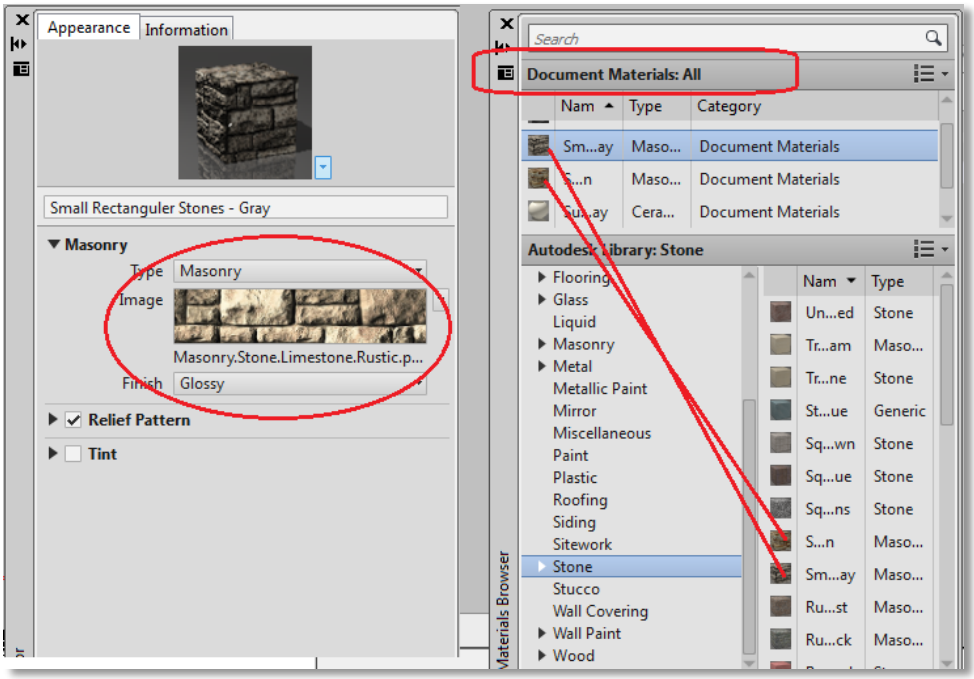
Apply Material to object

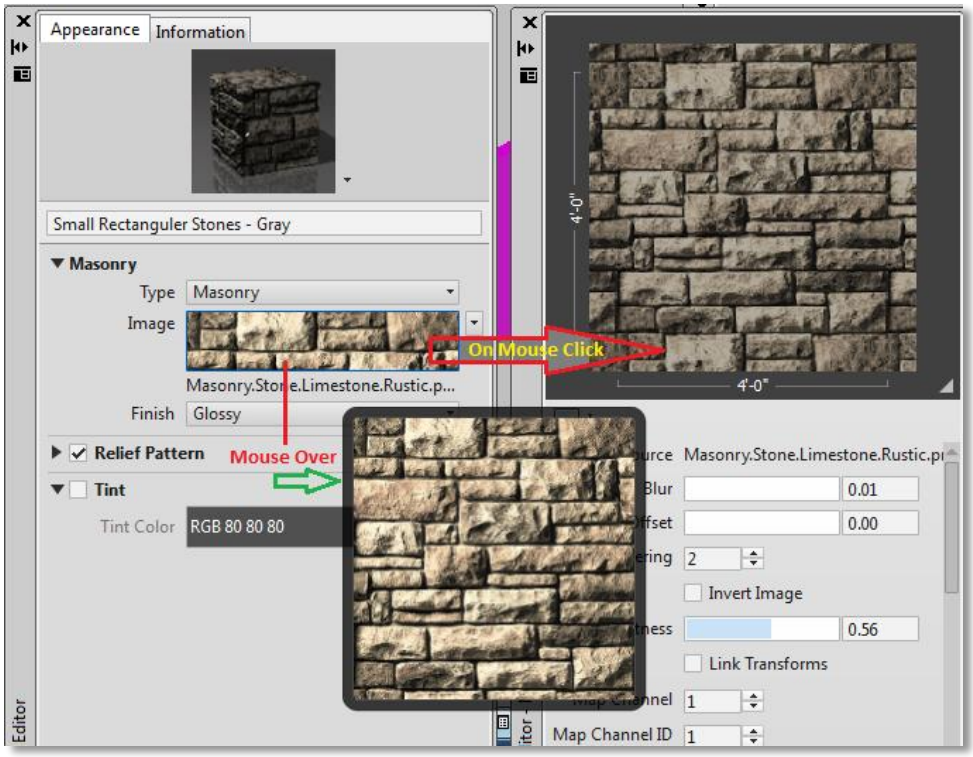
Apply Material to the following object

Step-1: Click Material→Browse Material

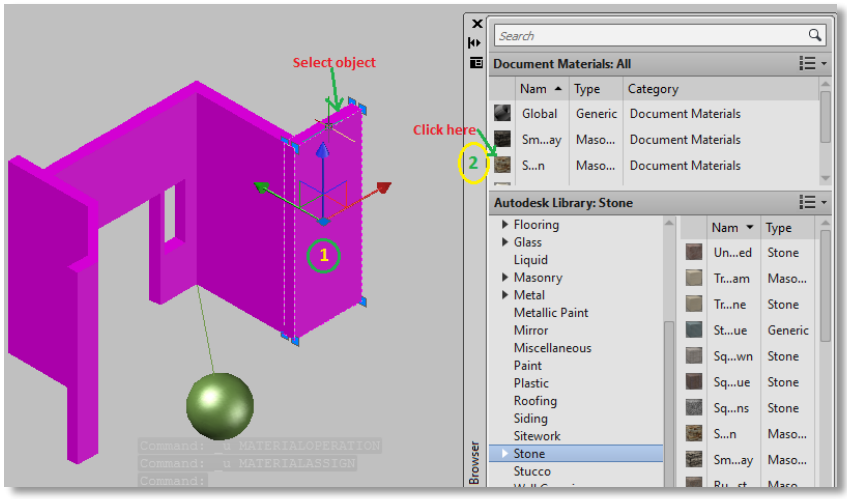


Step-2: Double Click or drag any of the material from material library into the document material list as shown in the figure below



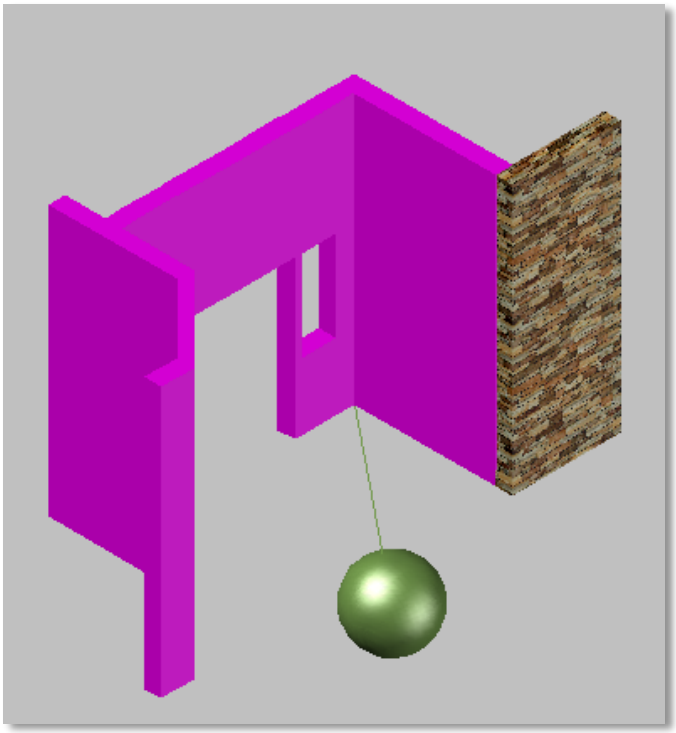


Step-3: Go to the Material list and apply material as shown in the figure



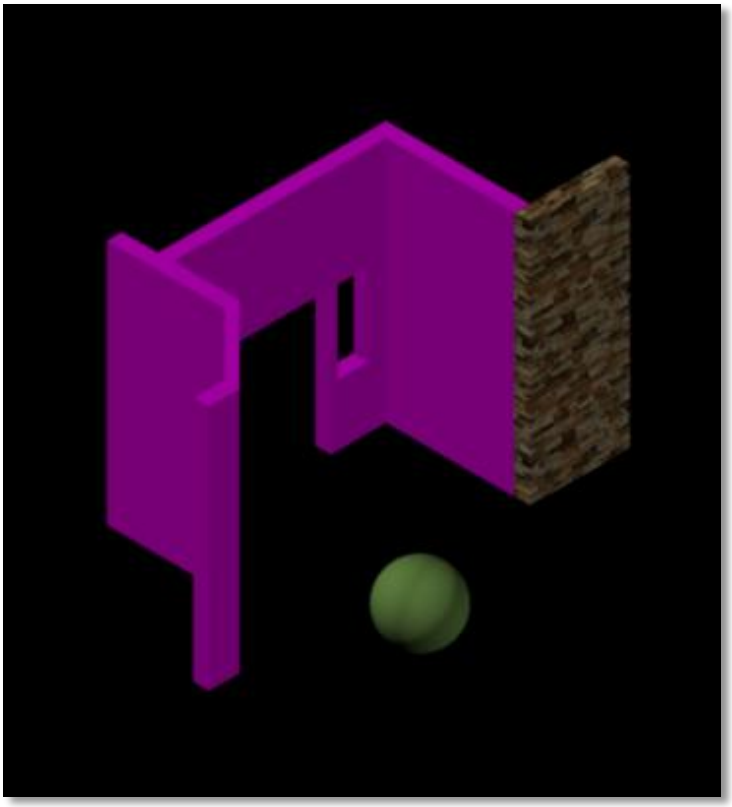
- You will get the following.
- Blank space Mention following

Note:
Visual Styles Control must be in realistic view.

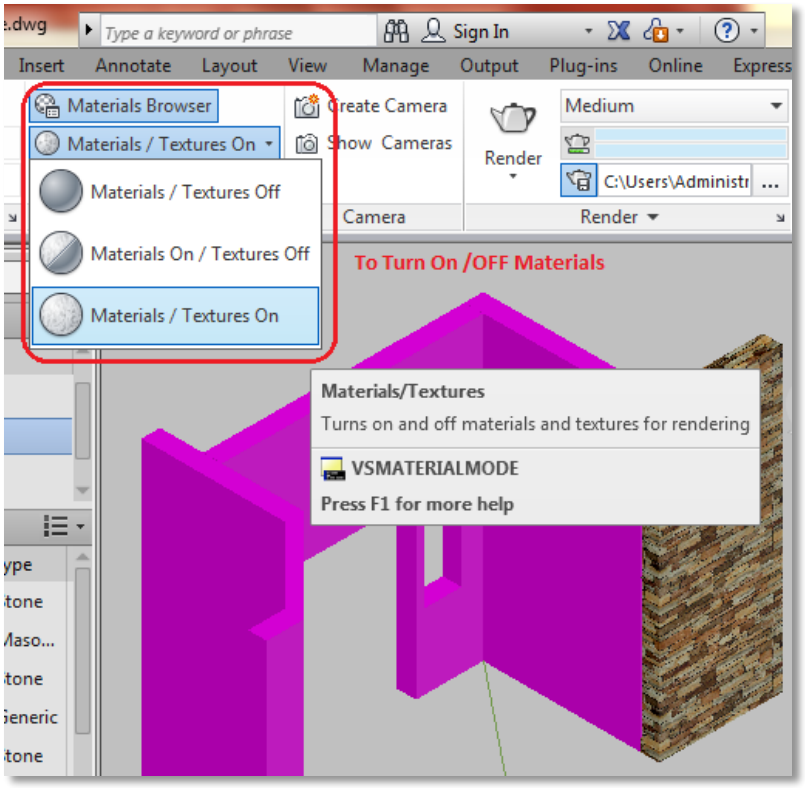


Step-4: Type **render** command in command bar to view the render with applied material.

Past picture here



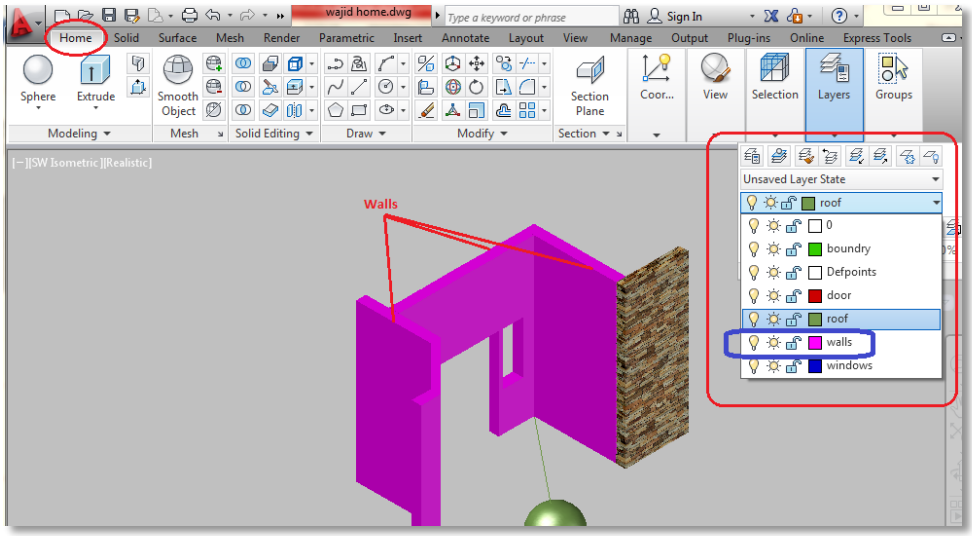
- To Turn ON /OFF Material



2. Material by Layers

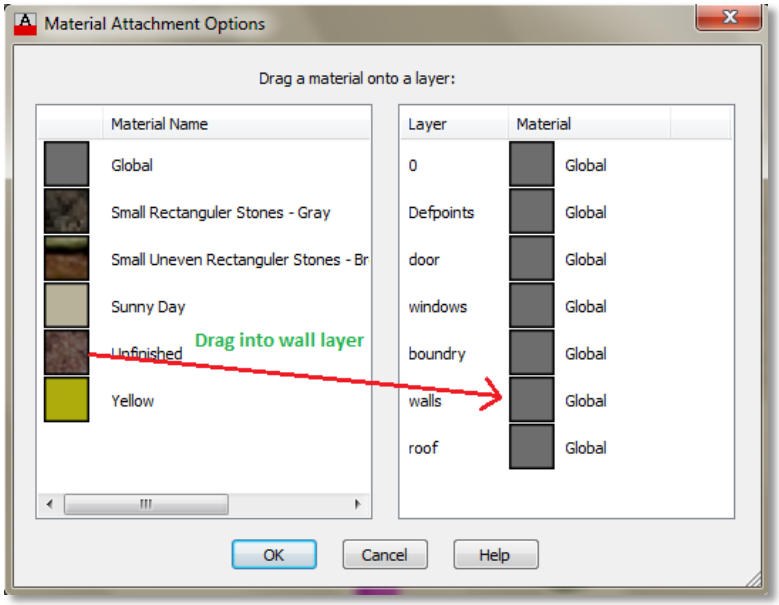
You can apply material by layer but the layer must define before apply **"Material by layer"**.

In this example we have already define layer for the drawing as shown in the figure below.

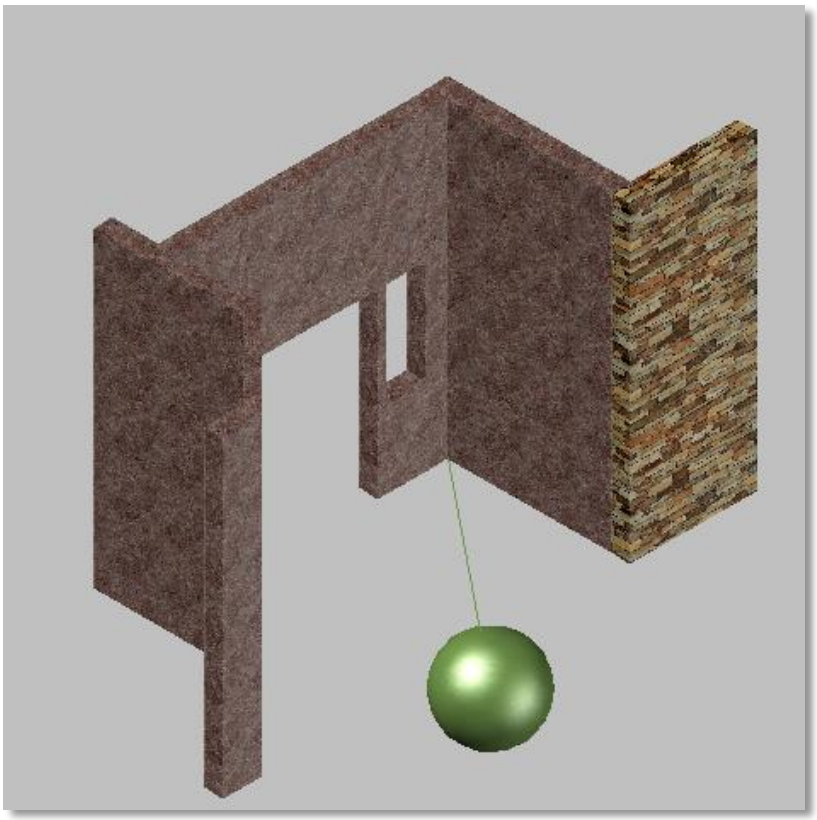


Step-1: You can attach a material to an entire layer using the Material Attach command.

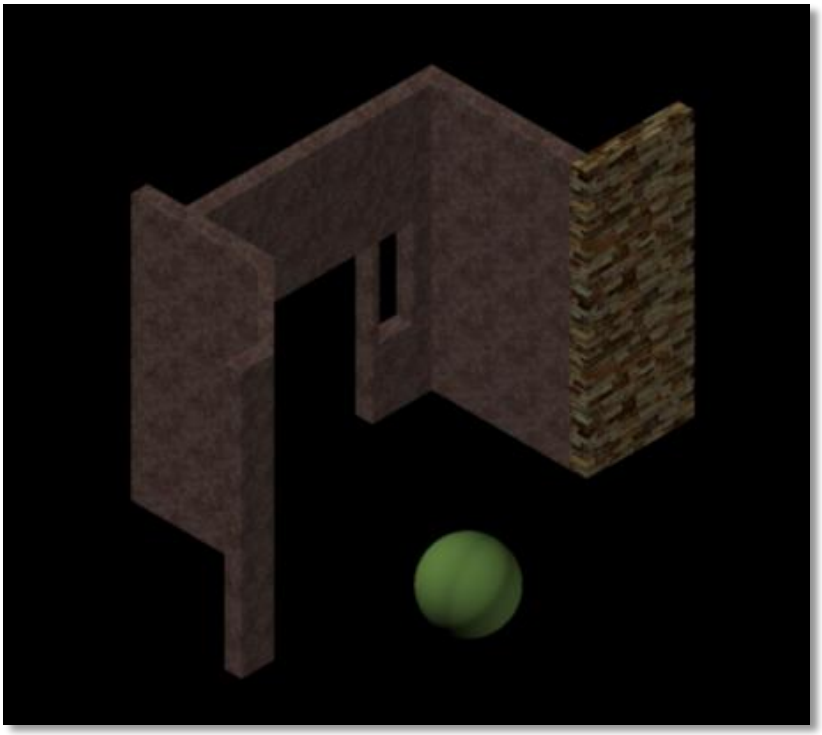
- Type **MATERIALATTACH** command in command bar <Enter>



Output should be as follow



Step-2: Apply render by render command.

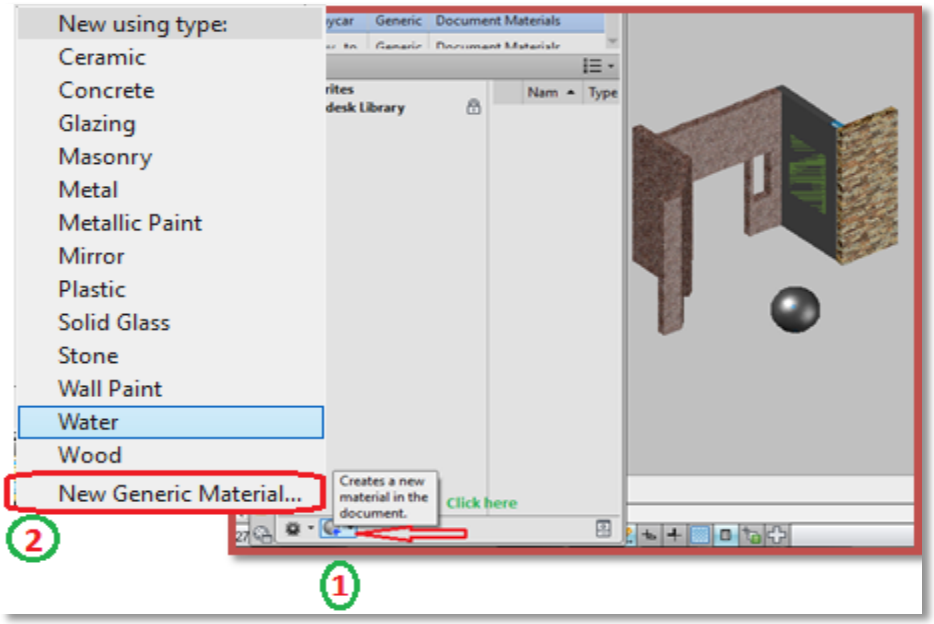


3. Create own Material

You can create your own material to apply photo and images.

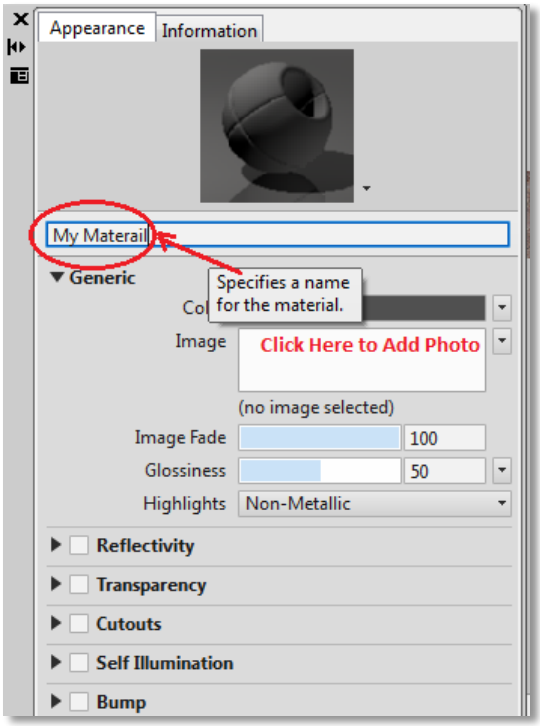
Step-1: Type materials command in command bar or click Browse Material

Step-2: Click Create **New Material** Option as shown in the figure below.

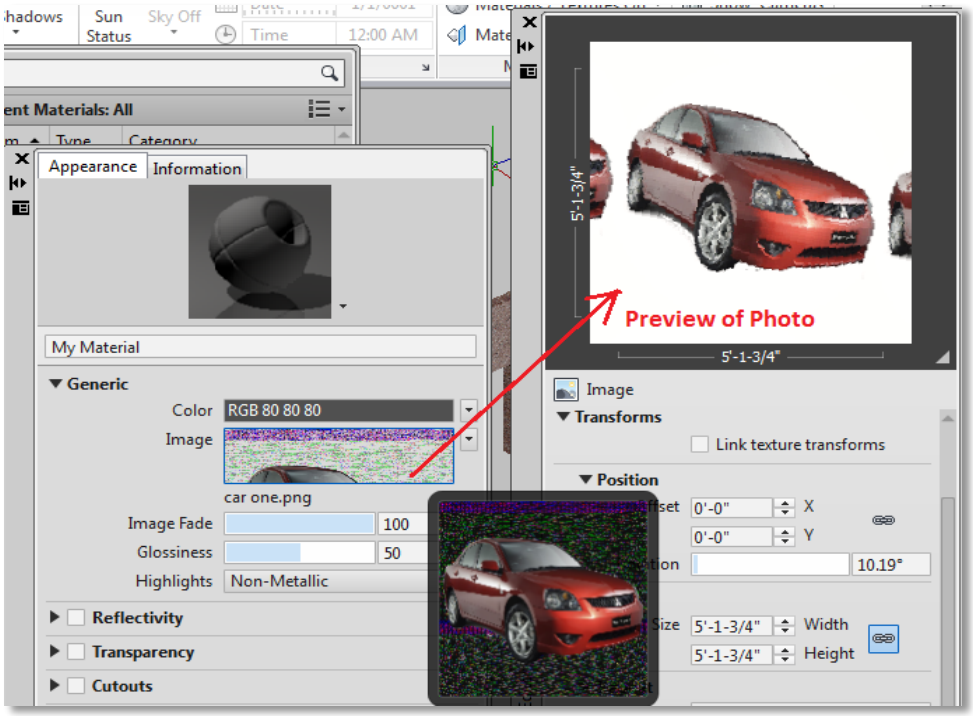


Step-3: Click a **New Generic Material** and then follow steps as described in the figure below

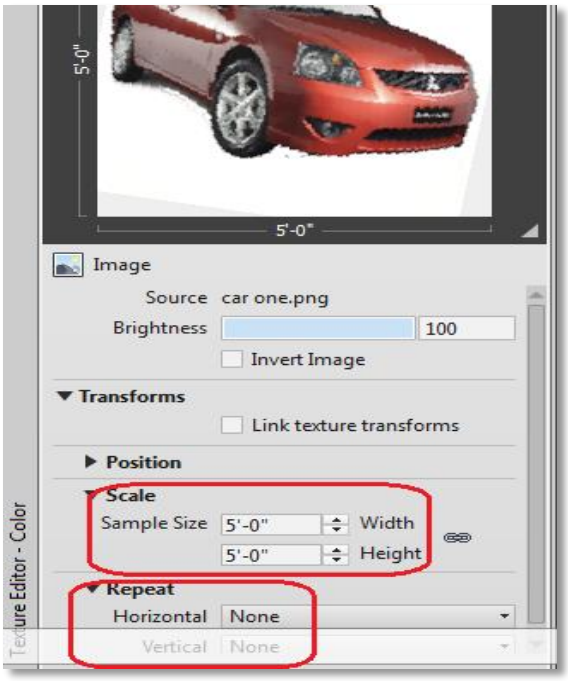
Write down step

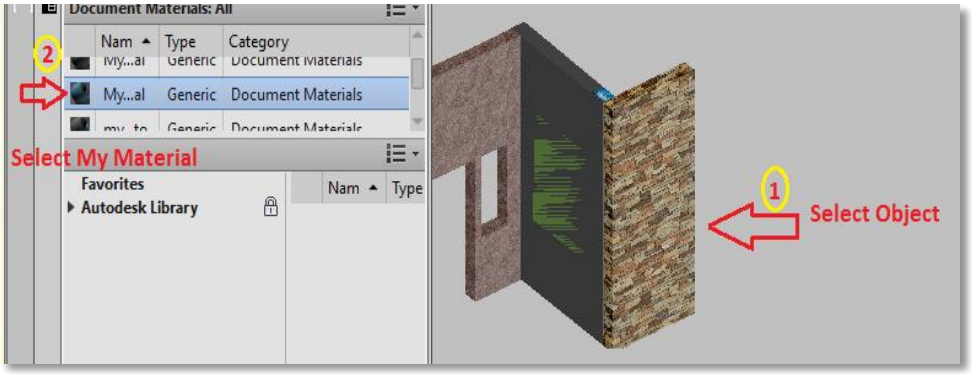


Step-4: Browse to file for adding any Photo as shown in the figure below.

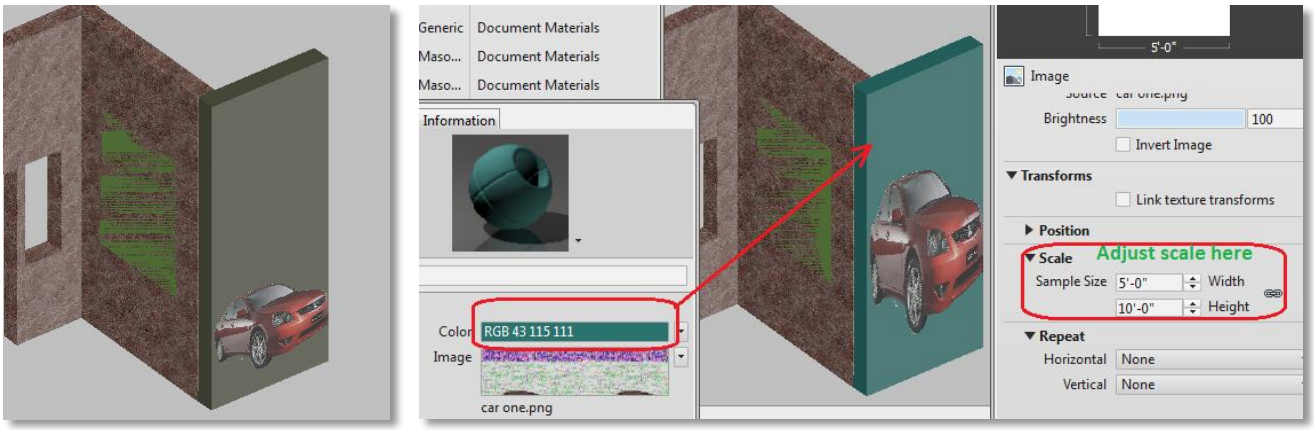


Step-5: Apply material to the selected object.

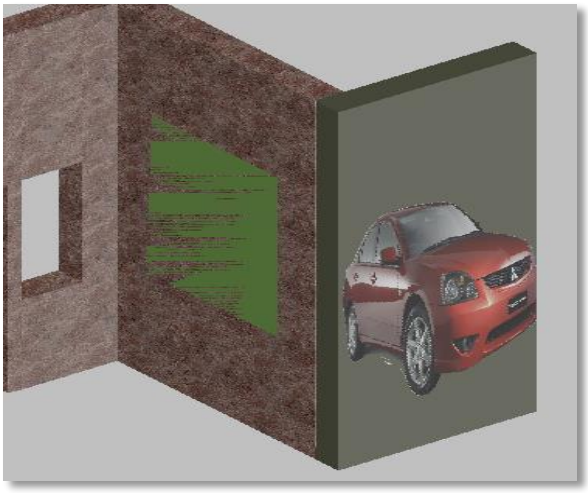




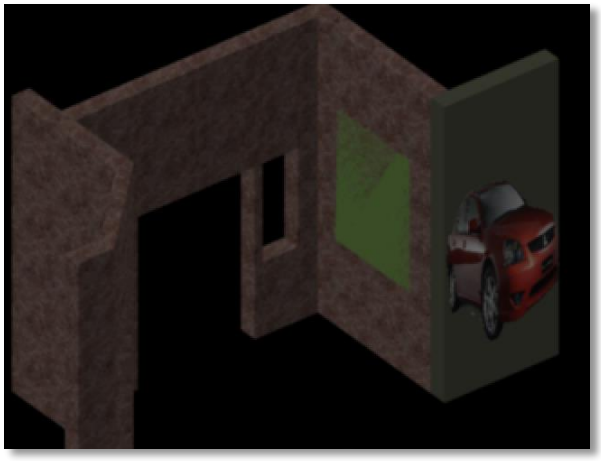
Compose the papers



Final Out Put



Step-6: Apply render command to display render



4. Material Mapping

You can map material to an object by using **"MATERIALMAP"** command.

Step-1: Type **MATERIALMAP** command in command bar <Enter>

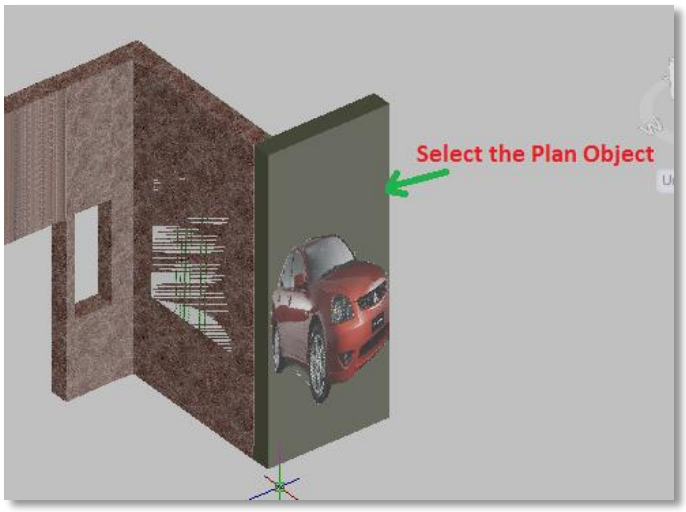
Step-2: Select an option [Box/Planar/Spherical/Cylindrical/copy mapping to/Reset mapping]<Box>: **p**

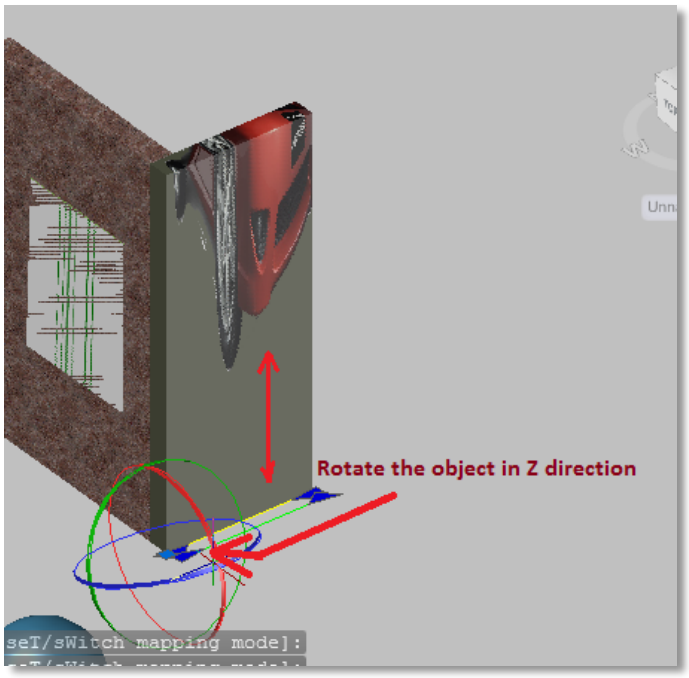
Step-3: Select faces or objects: 1 found

Step-4: Select faces or objects: <Enter>

Step-5: Accept the mapping or [Move/Rotate/reseT/sWitch mapping mode]: **R enter**

Step-6: Rotate the selected face or object by **GIZMO** to map the material applied.





Learner's Activity

Apply material Mapping to map the Face and Object by using the following options.

- BOX
- SPHERE
- CYLANDER
- COPY

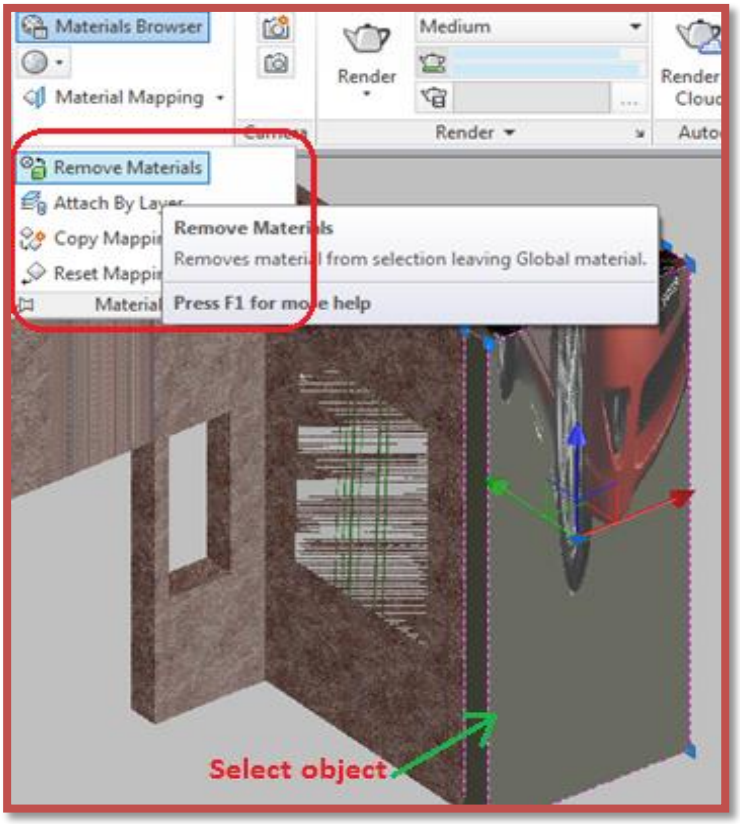
5. Remove Material

You can remove Material from an object by using the following method.

Method:

Select the object

Go through 3D Modeling→Render tab→Material→Remove Material



6. Dump Map

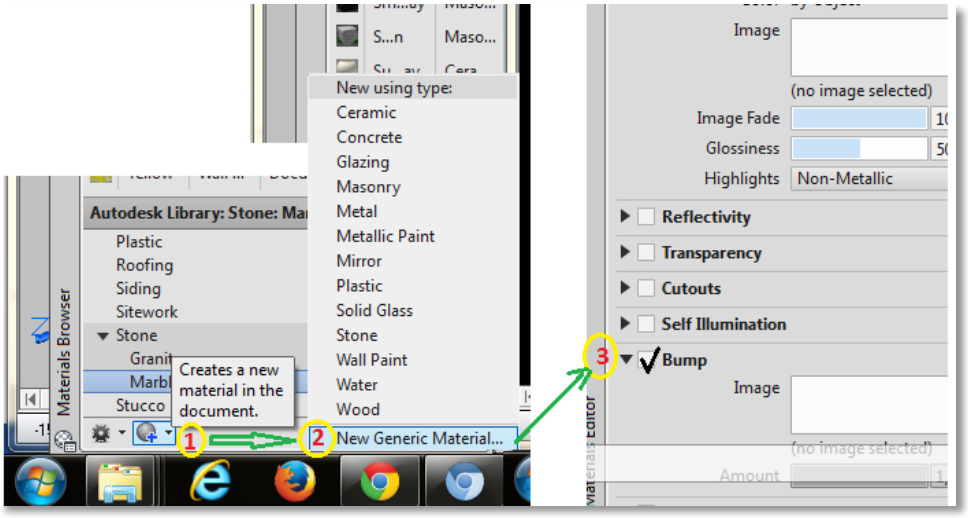
The **Bump** option contains settings for making some areas of the material appear raised and other areas depressed.

Dump map is the same as creating new material and Material Mapping option, so this topic is left for the learner as learner activity to apply dump map.

Learner's Activity

You can follow the following figure to apply dump map.

Hints:



7. Materials Turn ON/OFF

You can execute **"VSMATERIALMODE"** command to On/Off Materials by two method.

Method-1:

Step-1: Type VSMATERIALMODE command in command bar <ENTER>

Step-2: Follow the options

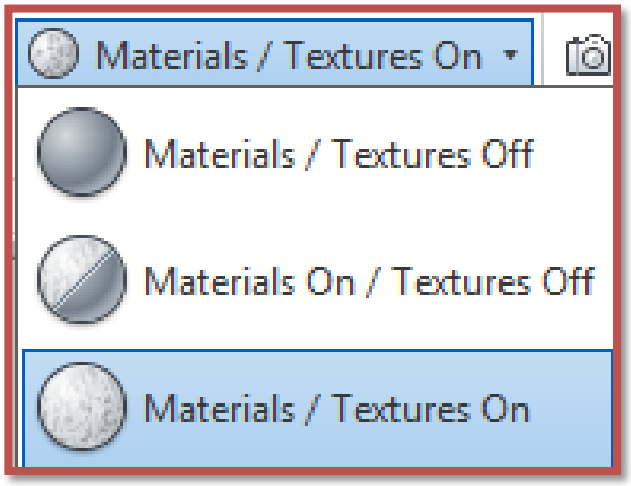
- Enter new value for VSMATERIALMODE <2>: **1** <For Turn Off>

Or

- Enter new value for VSMATERIALMODE <1>: 2 <For Turn On>

Method-2:

Go through 3D Modeling→render tab→



Learning Unit 3: Apply Lights

Overview

Without a lighting scheme, the scene may be look unattractive. In this unit, you will able to add a light effect to represent the sun and then an additional light to add ambient illumination to the scene.

You will also be able to differentiate between the functionality of different types of light and its implementation in the drawing and get understanding of geographical location setting by specifying the latitude and longitude of a location for the sunlight.

1. Light

As we know that without a light, the scene cannot be looked attractive and realistic.

Rendering has four conceptual steps to develop drawing more realistic.

- Create the model
- Attach materials to objects
- Render the image
- Place lights

We already discussed the first three steps of rendering "**Creating the model, Attach material** and then **Render the image or drawing"**, the last conceptual step **Place lights** will we covered in this unit.

2. Creating a Light Source

AutoCAD has four kinds of lighting, each with a distinct method for distributing light rays into the scene. They are as follows:

Point Light: All light rays are emitted from a single location and diverge as they get farther away. An incandescent light bulb is a real-world example of a point light, even though the light does not travel in the direction of the light's fixture.

Spotlight: With this type of light, rays are emitted from a single point, but they are restricted to a conical portion of the amount of light that a similar point light would emit. Flashlights and headlights are examples.

Distant Light: With this type of light, all light rays are parallel. Although the sun is technically a point light, at the enormous distance the light rays travel to Earth, they are nearly parallel.

Weblight: These are photometric lights with real-world distributions. These lights can be used in conjunction with light distributions derived by manufacturers of real-world lights. Using manufacturer data to establish lighting distributions helps ensure more accurate representation of rendered lights than possible when using point or spotlights.

3. Default Lighting

When there are no lights in a scene, the scene is shaded with default lighting. Default lighting is derived from two distant sources that follow the viewpoint as you move around the model. All faces in the model are illuminated so that they are visually discernible. You can control brightness and contrast, but you do not need to create or place lights yourself.

When you insert custom lights or add sunlight, you can disable the default lighting. You can apply default lighting to the viewport only; at the same time, you can apply custom lights to the rendering.

4. Turn On/Off the default Lighting

You can turn On/Off the default lighting by using “DEFAULTLIGHTING” command as shown below

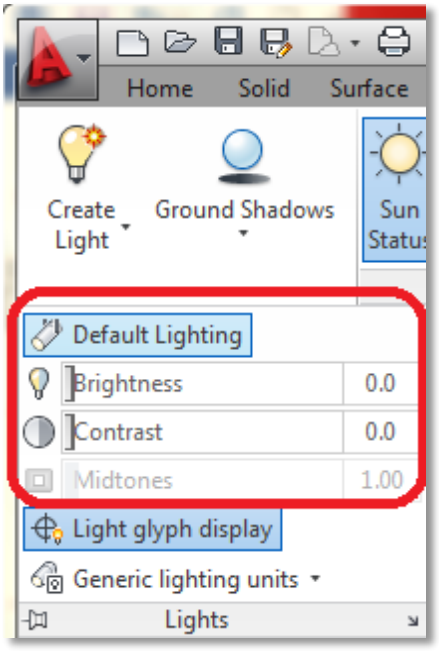
Method-1:

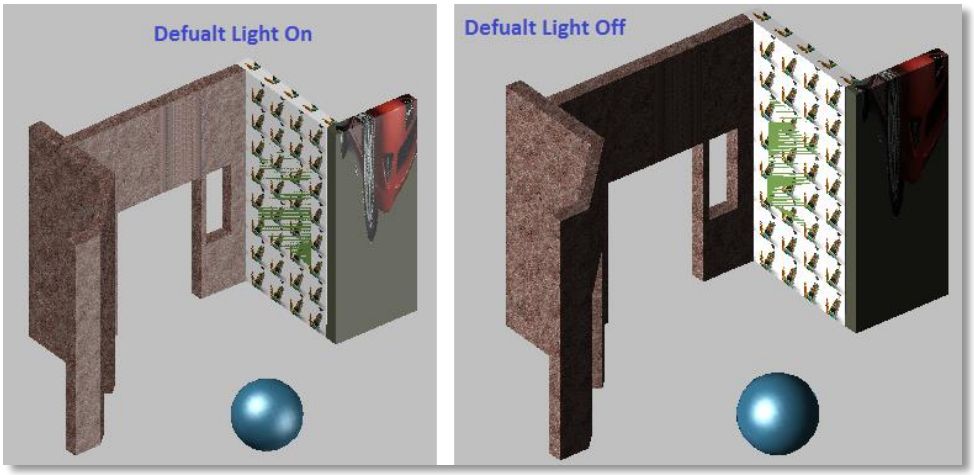
Step-1: Type DEFAULTLIGHTING command in command bar <Enter>

Step-2: Enter an option

- Enter new value for DEFAULTLIGHTING <0>: 1 <For On>
- Enter new value for DEFAULTLIGHTING <1>: 0 <For Off>

Method-2: Follow the figure below.



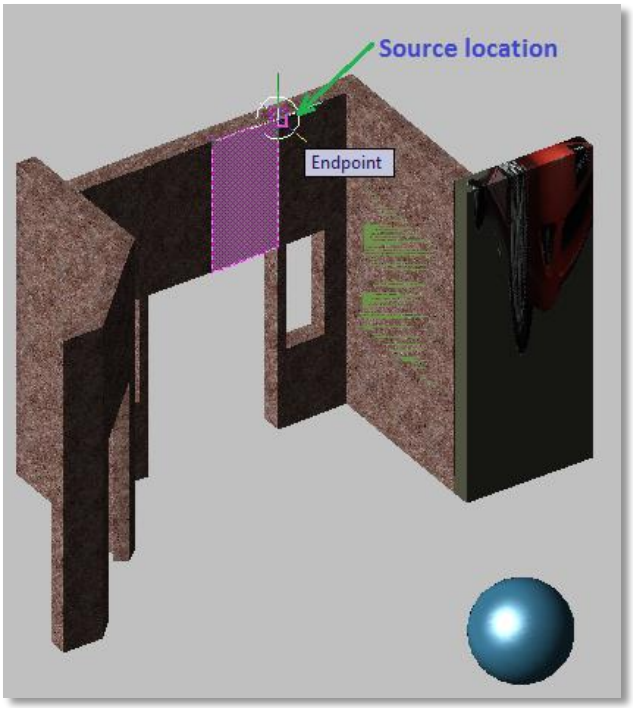


5. POINTLIGHT

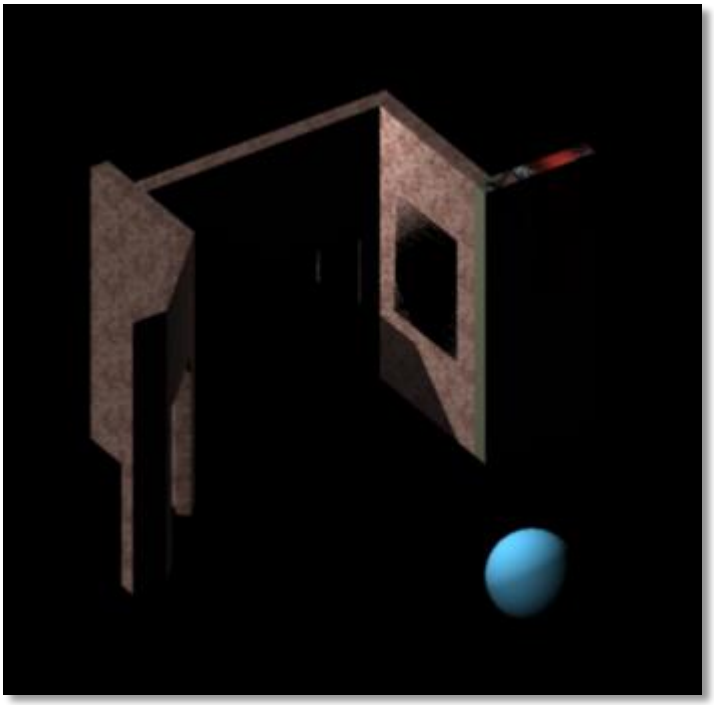
Point light that radiates light in all directions from its location.

Step-1: type **pointlight** command in command bar <Enter>
Or Click go through 3D Modelling→render tab→Light→PointLight

Step-2: Specify source location <0,0,0>: <Click or Define Source Location>

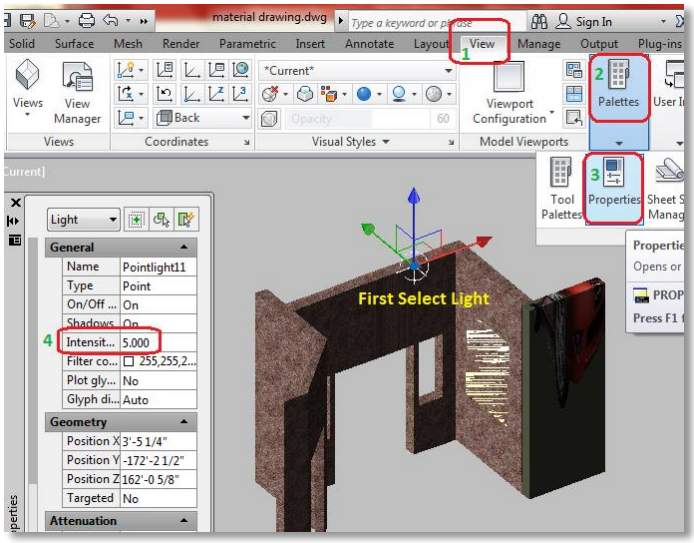


Step-3: Apply Rendering



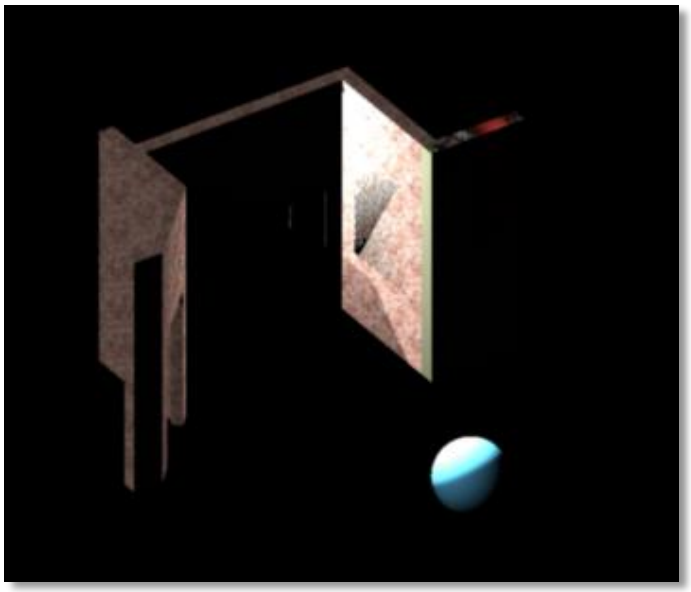
6. Photometric Lights

You can control over lighting, you can use photometric lights to illuminate your model.
To change the intensity of light, you should follow the following figure step by step.



Note:
You can also change the color by using the filter color option.

Apply Rendering



Learner’s Activity

Customize Photometric light energy

- light for lighting units
- Luminaries
- Weblight
- Halogen effect
- Candela intensity

7. SPOTLIGHT

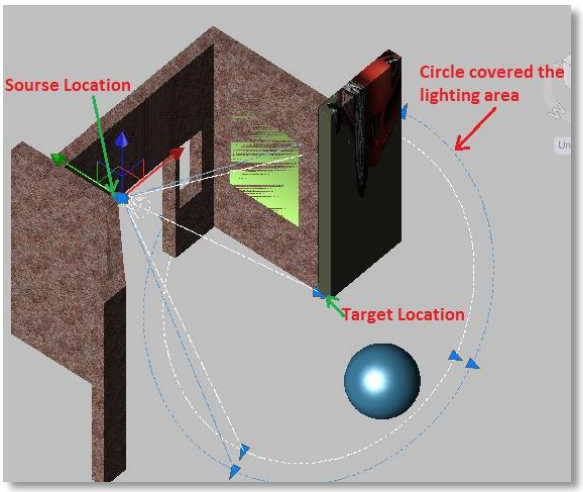
Spot Light emits a directional cone of light.

Step-1: type **spotlight** command in command bar <Enter>

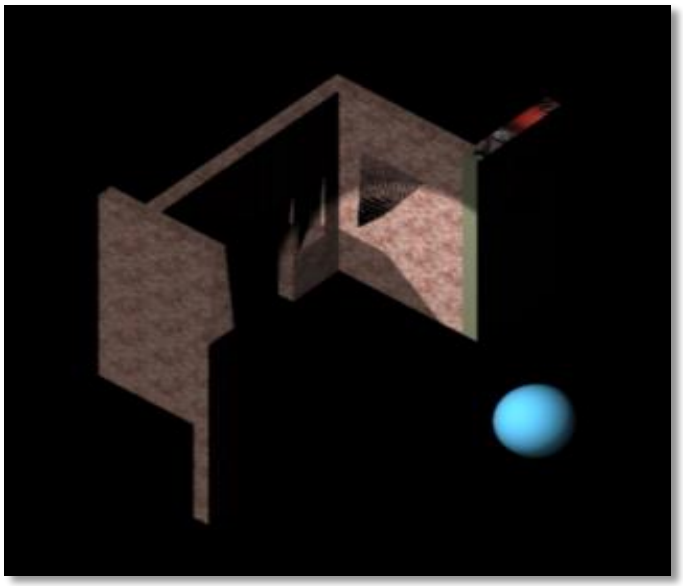
Or Click go through 3D Modelling→render tab→Light→SpotLight

Step-2: Specify source location <0,0,0>: <Click or Define Source Location>

Step-3: Specify target location <0,0,0>: <Click or Define Source Location>



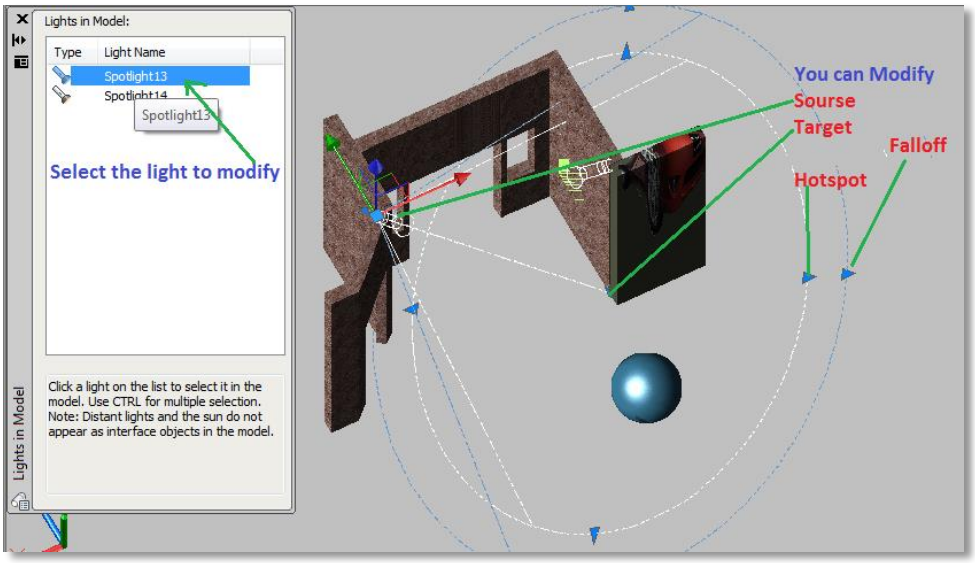
Step-4: Apply rendering



8. Modification of Lights

You can modify in light target location and area to cover by using LIGHTLIST command.

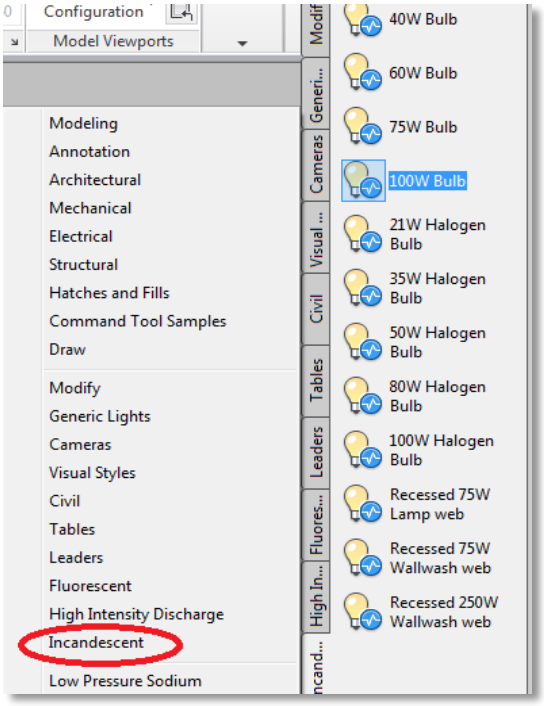
Type LIGHTLIST command in command bar <Enter> you will get the following as shown in the figure below



9. Lights Tool Palette

Default lights can be added to your drawing from the Tool Palette menus.

- Press Ctrl + 3 From Keyboard to display Light tool palette



10. DISTANTLIGHT

Distance light is used for uniform parallel light rays in one direction and mentioning from and to points.

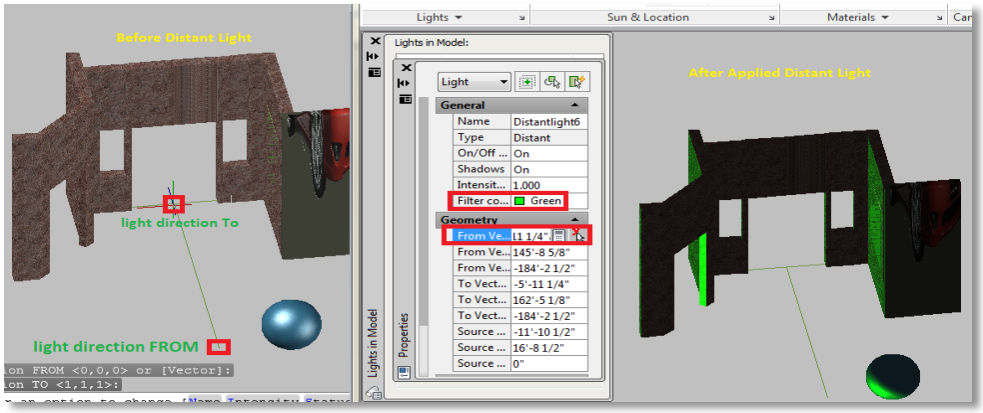
Step-1: type **distantlight** command in command bar <Enter>

Or Click go through 3D Modelling→render tab→Light→DistantLight

Step-2: Specify light direction FROM <0,0,0> or [Vector]:: <Click or Define Source Location>

Step-3: Specify light direction TO <1,1,1>: <Click or Define Source Location>

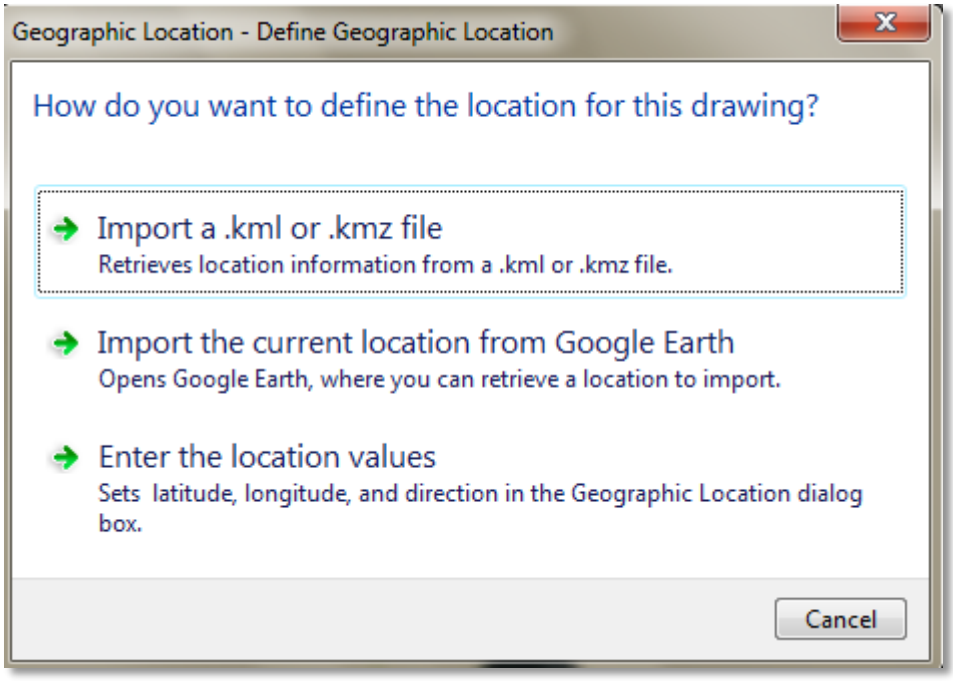
Step-4: Enter an option to change [Name/Intensity/Status/shadoW/Color/eXit] <eXit>: **<Enter>**



11. GEOGRAPHICLOCATION

Incorporate natural light based on climate into the drawing by specifying the latitude and longitude of a location for the sunlight using “GEOGRAPHICLOCATION” command.

For specifying these location we will face three options as shown in the figure below.



- **KMZ File**

KMZ is a file extension for a place mark file used by Google Earth. KMZ stands for Keyhole Markup language Zipped. It is a compressed version of a **KML** (Keyhole Markup Language) file. Keyhole was the founding company of the Earth Viewer software that Google Earth was built upon.

KMZ files can contain place mark featuring a custom name; the latitudinal and longitudinal coordinates for the location, and 3D model data. KMZ files can be opened by Google Earth, or **unzipped** with a compression utility, such as **WinZip** on Windows, MacZip for Macintosh users, and Zip and UnZip for UNIX systems.

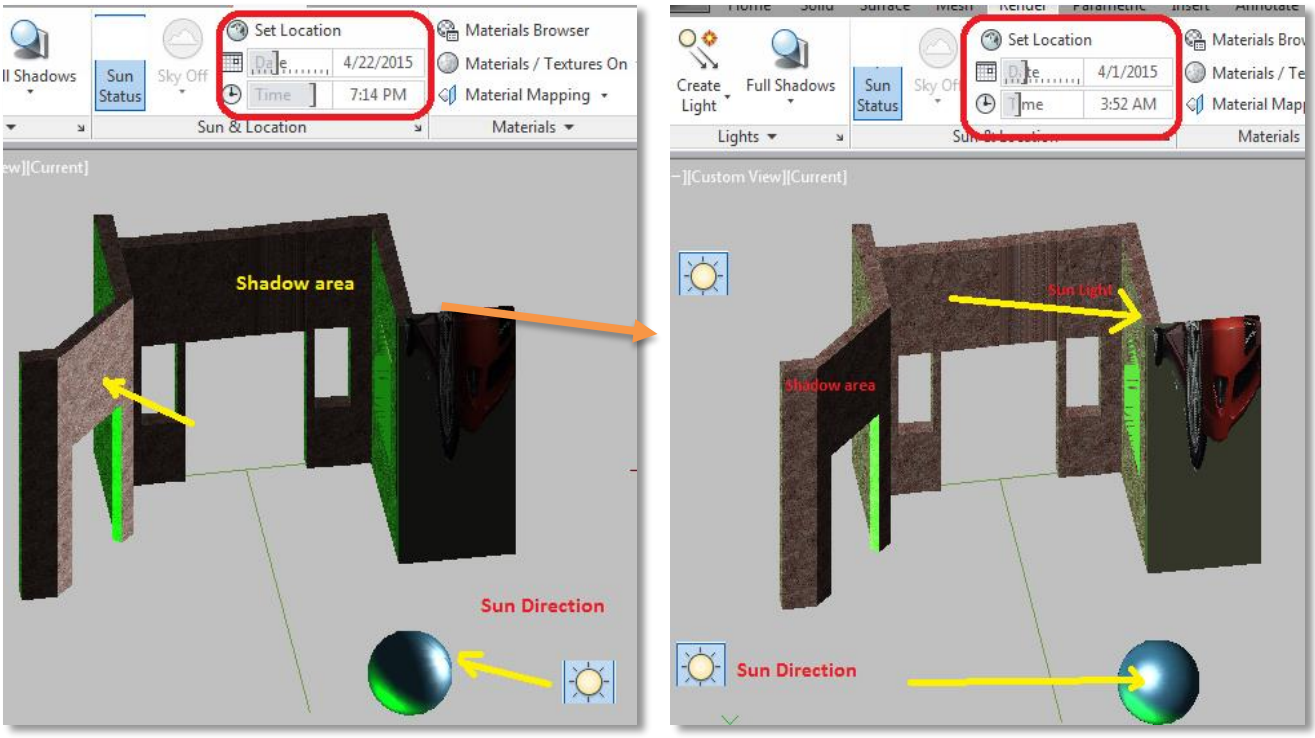
- **Import the Current Location from Google Earth**

Retrieve location information from a specific location in Google Earth. **Google Earth** should be installed and open with the location selected.

- **Enter the Location Values**

Display the Geographic Location dialog box, where you can enter the location information manually.

After setting the location you can set the time and date for the sunlight as shown in the figure below.


Set Location
Date 4/22/2015
Time 7:14 PM
Materials Browser
Materials / Textures On
Material Mapping
Sun Status
Sky Off
Sun & Location
Materials
Shadow area
Sun Direction
Create Light
Full Shadows
Lights
Set Location
Date 4/1/2015
Time 3:52 AM
Sun Light
Shadow area
Sun Direction

Summary of Module

- In this module we covered four conceptual steps (Create the model Attach materials to objects, Render the image, Place lights) of rendering, in three different units. In the very first unit we learnt the rendering techniques applied to an objects and drawing with different features of rendering destination, quality, selection, crop, and file by using **RPREF** command with environmental features to up atmospheric effects.

- In the second unit of this module we learnt how to apply material to an object in drawing, to make the appearance of the object more realistic and applied various materials such as wood, marble, brick and so on by using the material library of AutoCAD software material library. We also covered material mapping to the objects and saw full effects by rendered the scene.

- In the last unit of this module we covered the lighting techniques, as we see in our real world environment without a lighting the scene cannot look eye-catching and look flat. To make scene more realistic we covered light and its various types Point Light, Spot light and distant light and its geographical location setting with natural light setting.

Frequently Asked Questions (FAQs)

FAQ 1:	How could you get render crop?
Answer	You can get render crop by using rendercrop command or from 3D Modelling→render tab→render region.
FAQ 2:	How could you display rendered windows without rendering option?
Answer	You can get render windows from 3D-Modelling→render tab→render windows.
FAQ 3:	How could you switch between different rendered windows without rendering option?
Answer	Go through 3D-Modelling→render tab→render windows and switch between different render windows if you created with different setting.
FAQ 4:	Is there any option to off material applied to an objects by using RPREF command?
Answer	Yes! You can turn off material by using RPREF command in Materials option of Advance Rendering Properties.
FAQ 5:	How can you search for materials?
Answer	You can search for materials via search bar in Browse Materials dialog box

Test Yourself!

Please mark the correct one from the given options. You can check your answer with the Answer Key at the end of this module.

1. Which Visual Styles must be in ---------- view while you apply materials?

a. Conceptual View
b. Realistic View
c. Hidden View
d. None

2. Command for Advance Render Properties?

a. RPREF
b. RENDER
c. RPROP
d. Both b and c

3. Command for Materials Attach to an entire layer?

a. Materials
b. Materiallist
c. Materialattach
d. None

4. Command for material mapping?

a. MaterialMapping
b. Materialmap
c. Material
d. Materialist

5. Command used for Light list is ---------- ?

a. LIGHTLIST
b. LIST
c. LLIST
d. Both b and c

Answer Key

MCQ Number	Correct Answer
1	b
2	a
3	c
4	b
5	a

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