Engineering Fundamentals

ENG1100

UGNX – Drafting

Session Objectives

- Edge Operations
- Create Engineering Drawings in UGNX

- Student Presentations

Sketch Based Modeling

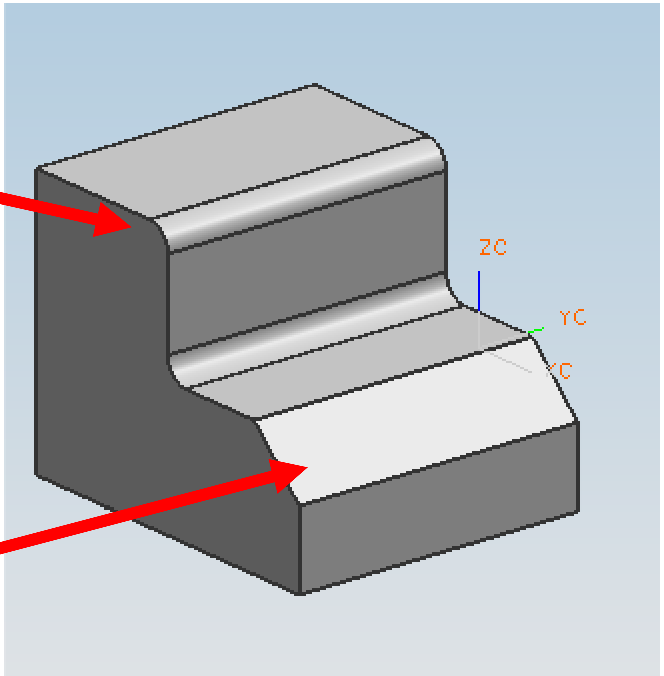
1. Sketch plane is chosen
2. 2-D sketch is created
   - Sketches can be the base feature of a solid body
   - Sketches can be added to an existing body
3. 3-D feature is created from the sketch (e.g. extrude, revolve … add, subtract, intersect)

Feature can be edited by editing the *constraints* of the sketch

Edge Operations

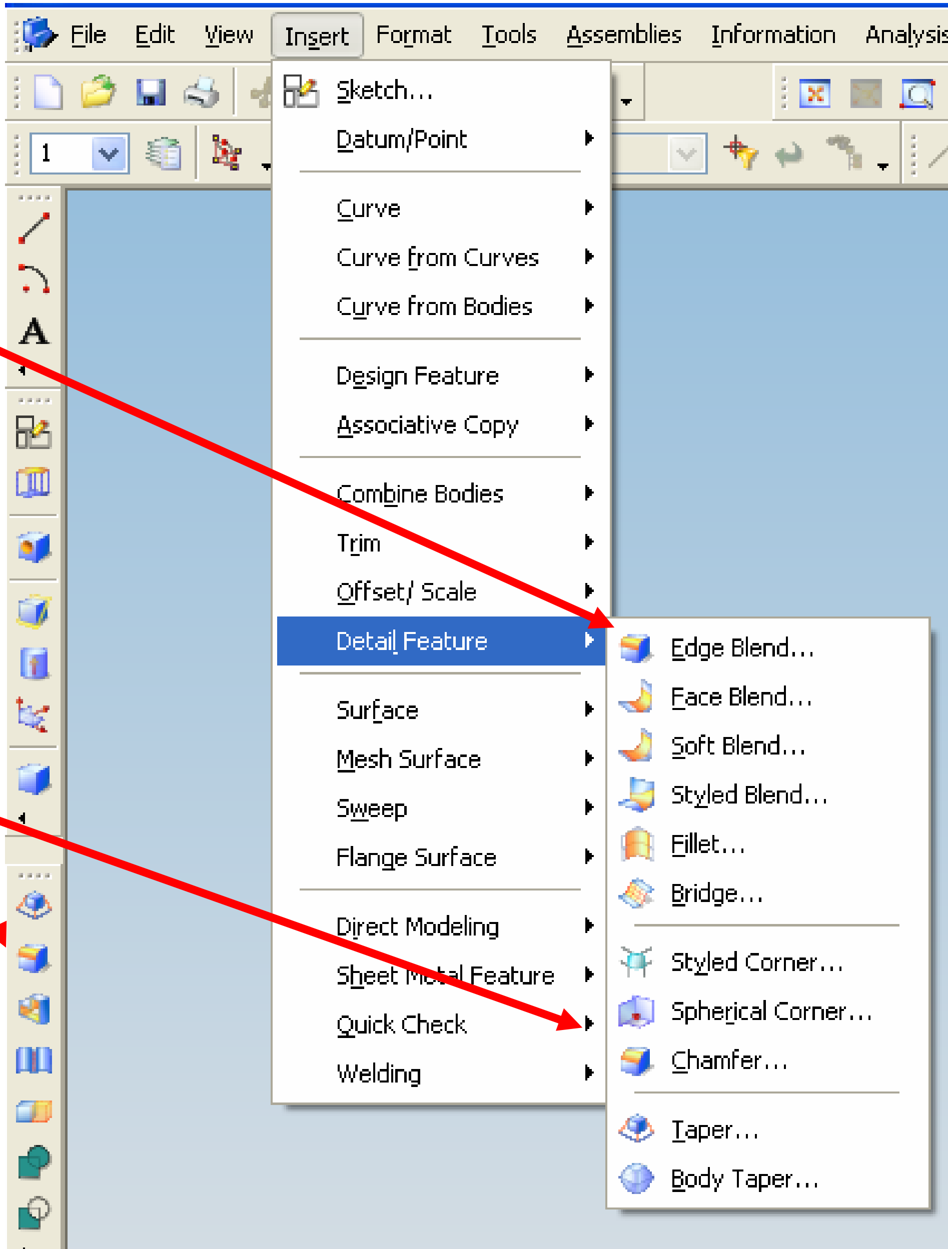
Can be added to model in modeling application

- <u>Edge blend</u>
  - Creates cylindrical faces in place of an edge
- <u>Chamfer</u>
  - Bevels the edges



Edge Operations

- Edge Blend
  - Select icon or Insert > Detail Feature > Edge Blend
  - Select edge(s)
  - Key in radius
- Chamfer
  - Select Insert > Detail Feature > Chamfer
  - Select edge(s)
  - Single offset
  - Key in offset



UGNX: Basic Drafting

Note: This is NOT a course in drafting, however, engineers need to be "*graphically literate*"; i.e., you need to be able to read and write engineering drawings.

Today's Objective:

- Learn basic process of converting a UG solid model into an Engineering Drawing

Software Structure

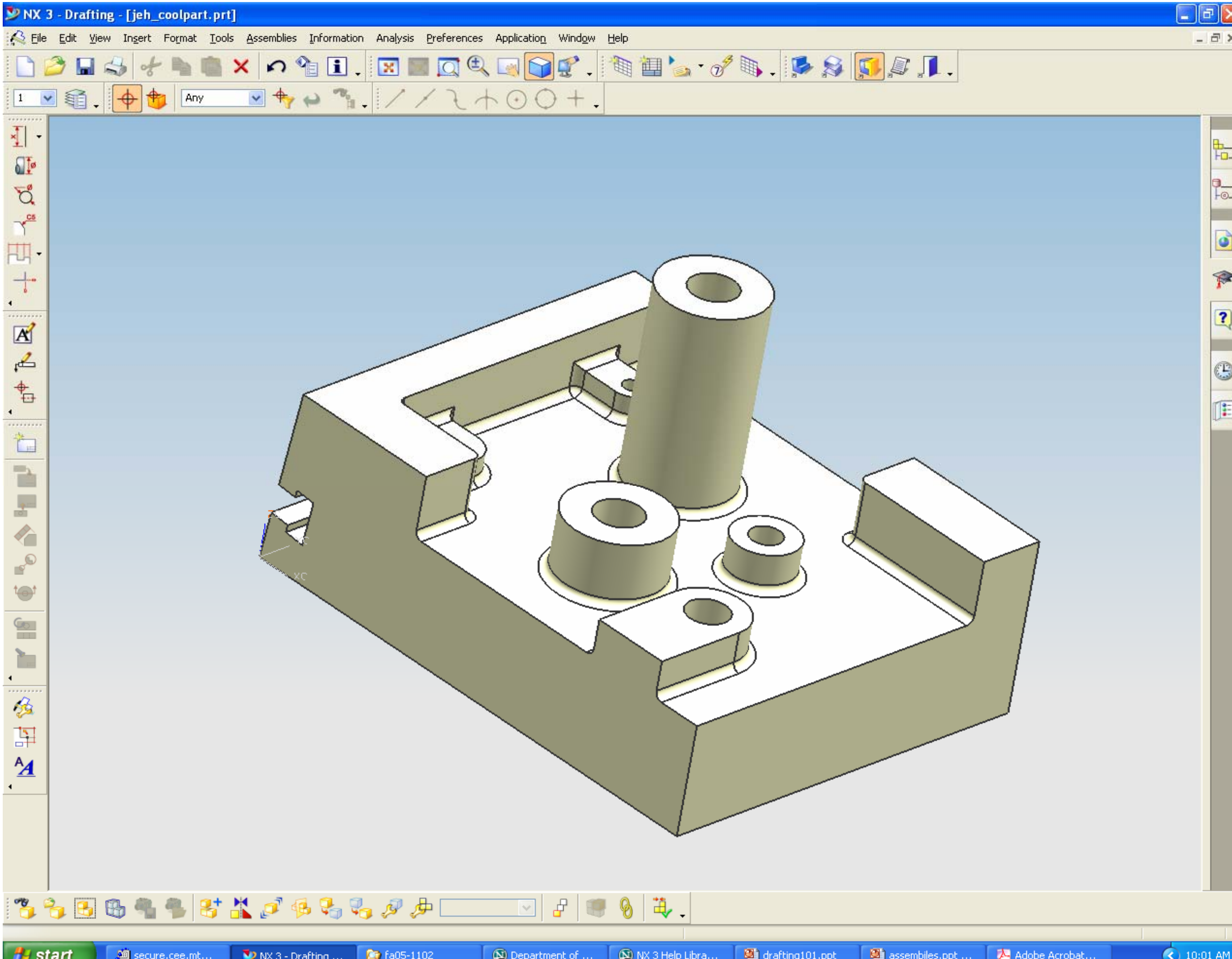
UGNX is organized into:

Applications: Gateway, Modeling, Drafting (and others)

When a part file is opened or created, the Gateway application is entered.

To create or edit a solid body the Modeling application is entered

To create an engineering drawing the Drafting application is entered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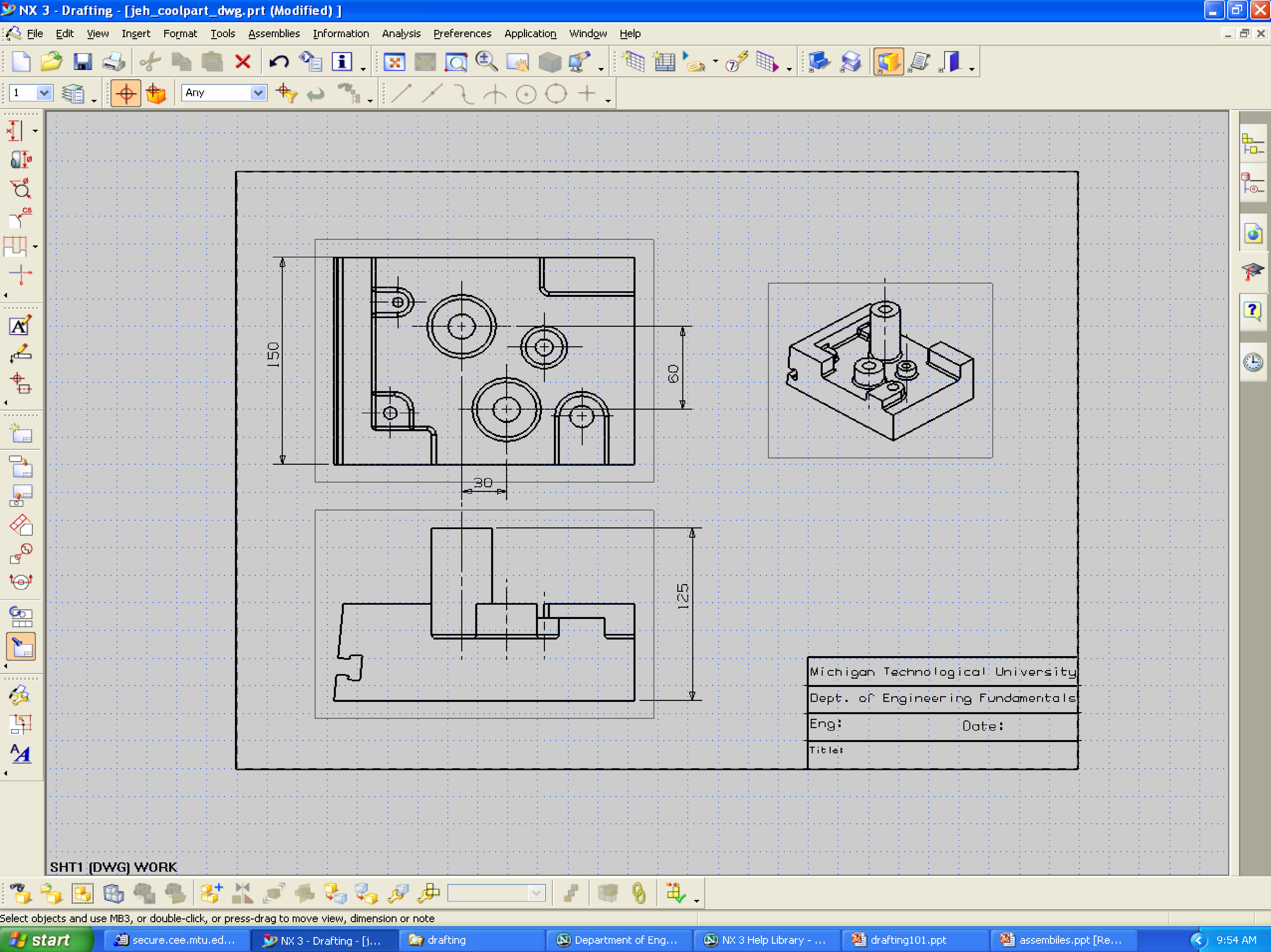
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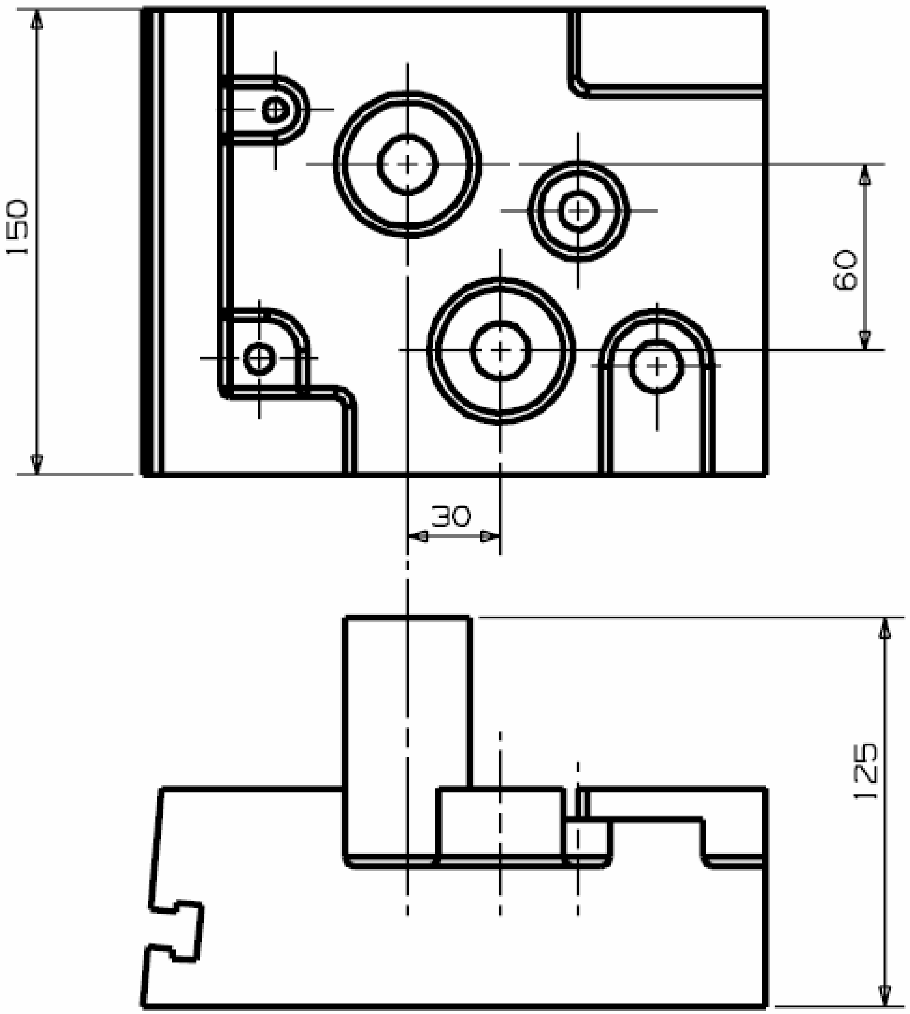
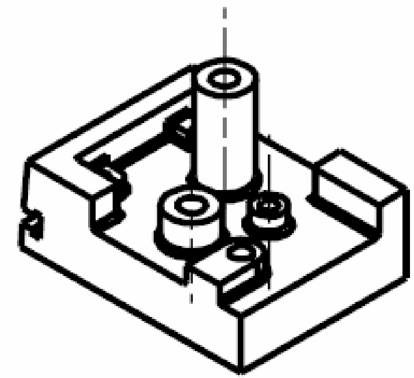




Michigan Technological University

Dept. of Engineering Fundamentals

Eng: Date:

Title:

UGNX: Basic Drafting

- The Drafting application lets you create drawings with views of the part, dimensions, and necessary drafting annotations.

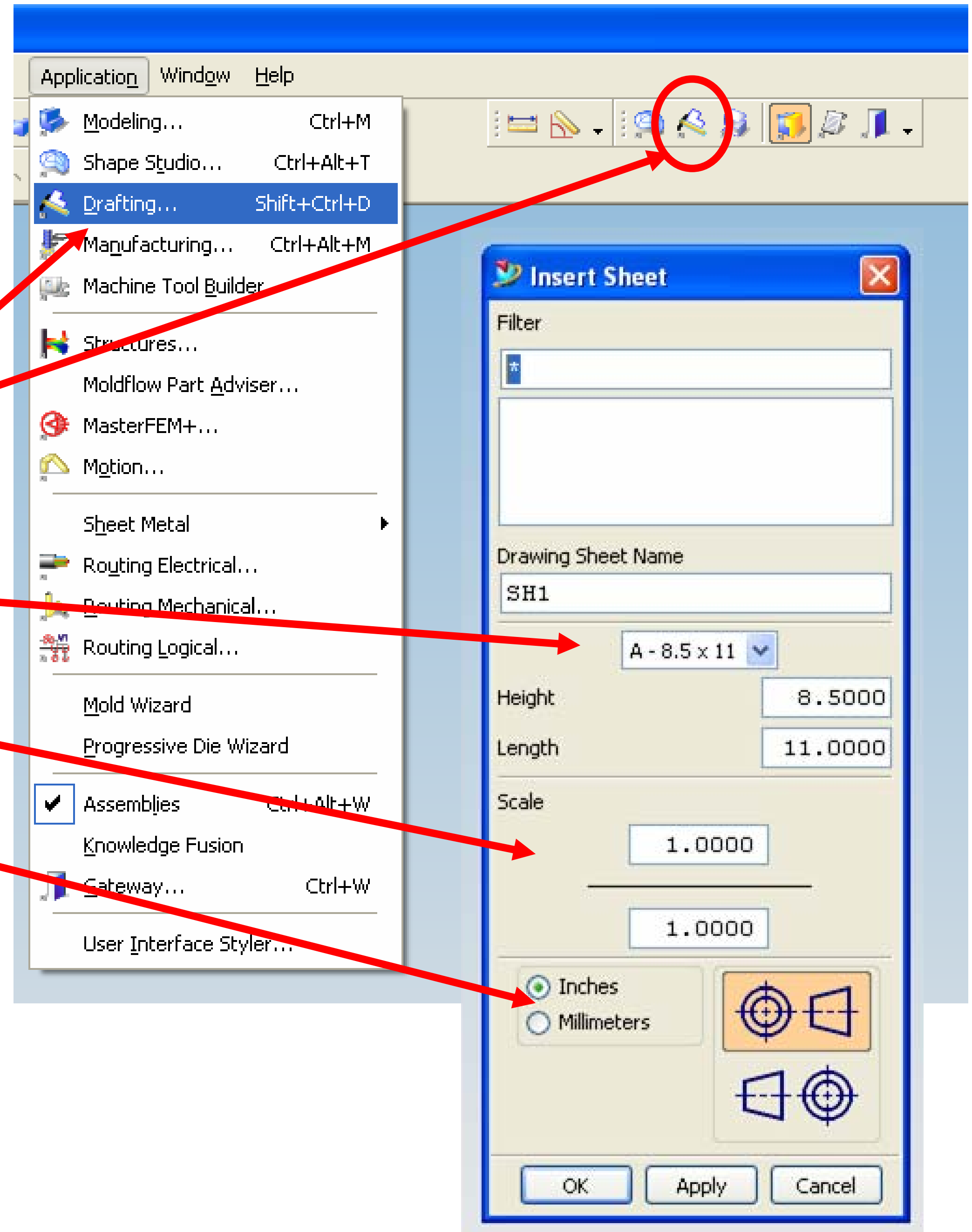
- This application supports the drafting of engineering models in accordance with ANSI and ISO standards.

Short Demo …

- Drafting Application
- Add Base View
    - Change view, style, orientation, scale
- Add Dimensions
- Add Annotation (text)

To Create a Drawing

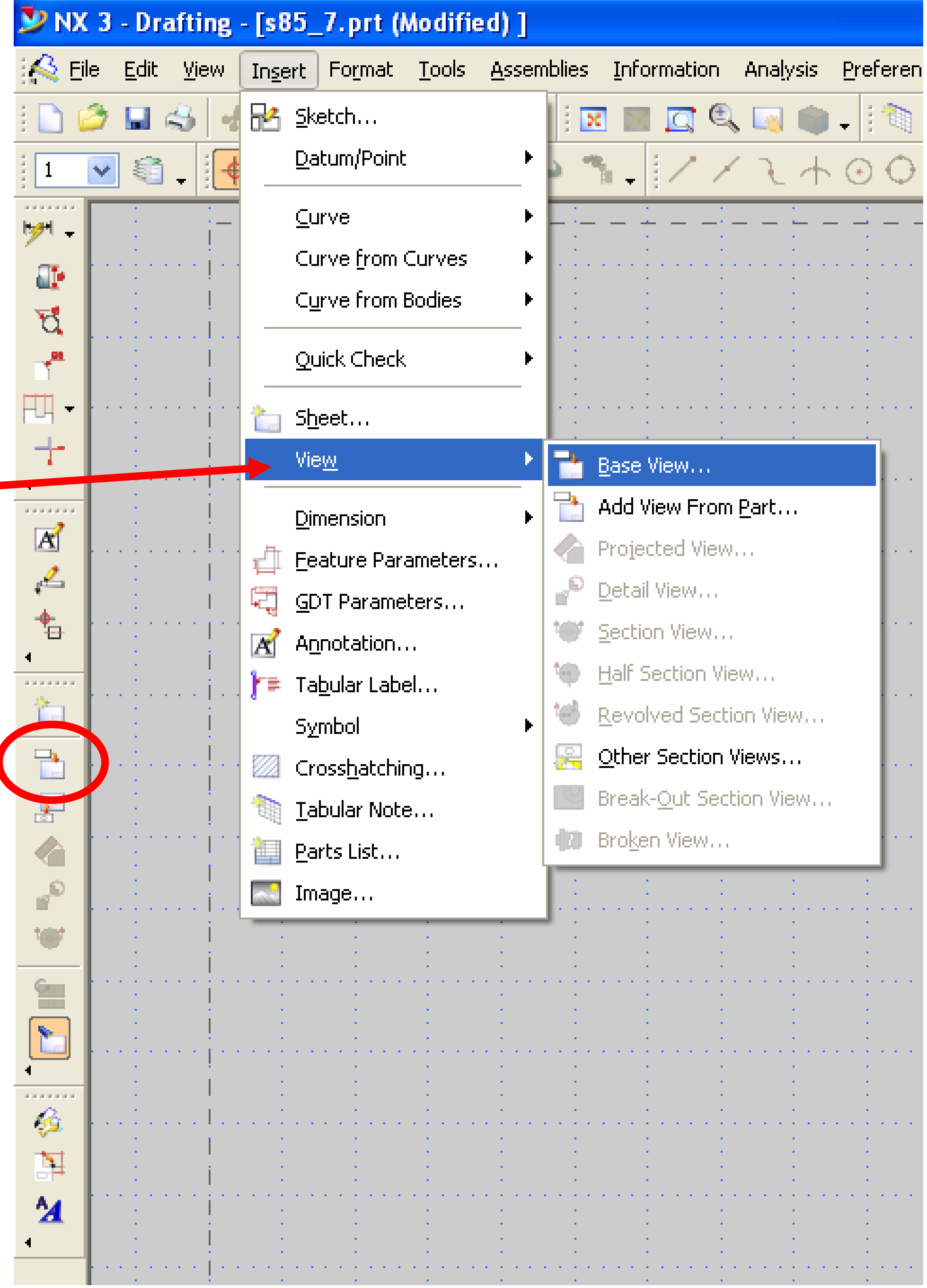
1. Create part in modeling application
2. Switch to drafting application
3. Insert Sheet form select:
   - Paper size
   - Scale
   - Units



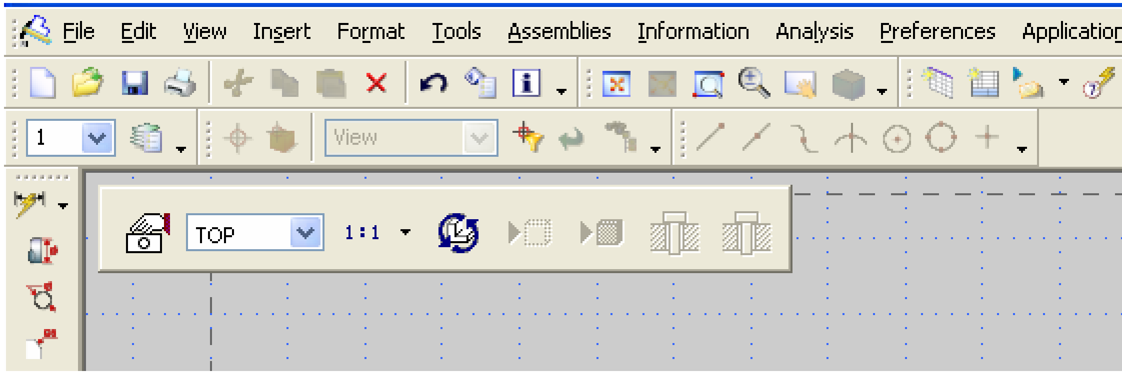
To Create a Drawing

4. Add Base View

- Insert > View > Base View
- MB3 on drawing border > Add Base View
- Icon on sidebar
- Part Nav MB3 on drawing sheet > Add Base View



To Create a Drawing

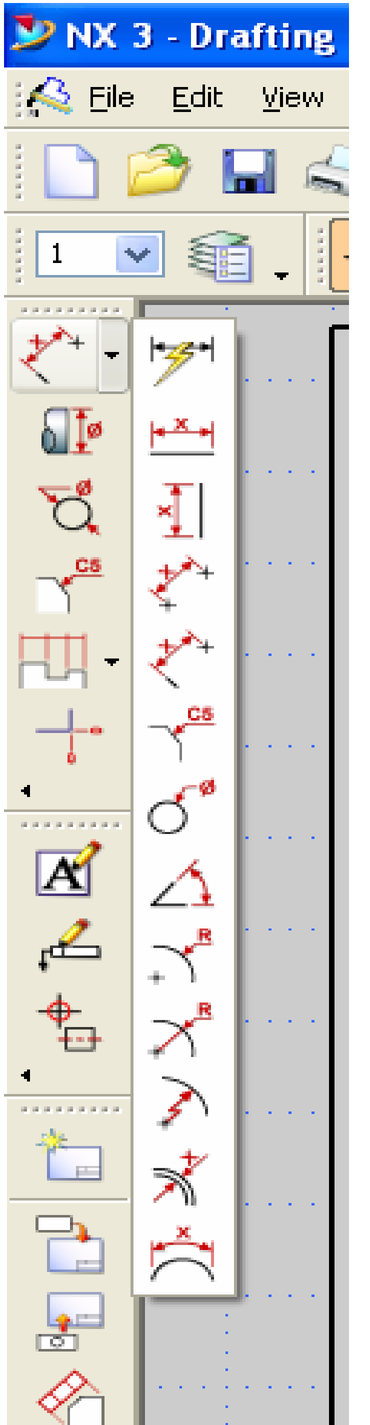


5. Choose View Options
   - Style (hidden lines)
   - View (front/iso)
   - Scale
6. Place Base View
7. Add Projected Views
8. To get an isometric, Add Base View

Adding Dimensions

- For many dimensions, just use the Inferred dimension tool
- To be more specific, pick the tool that best selects your part geometry

- Dimensions are fully *associated* to solid, i.e. if you change the model (in modeling) dimensions (in drafting) automatically update



Adding Notes (Annotation)

- Select Annotation Editor
  - Insert > Annotation
  - Icon
- Enter Text in Edit Window
- Place Text