

# CAD 2009

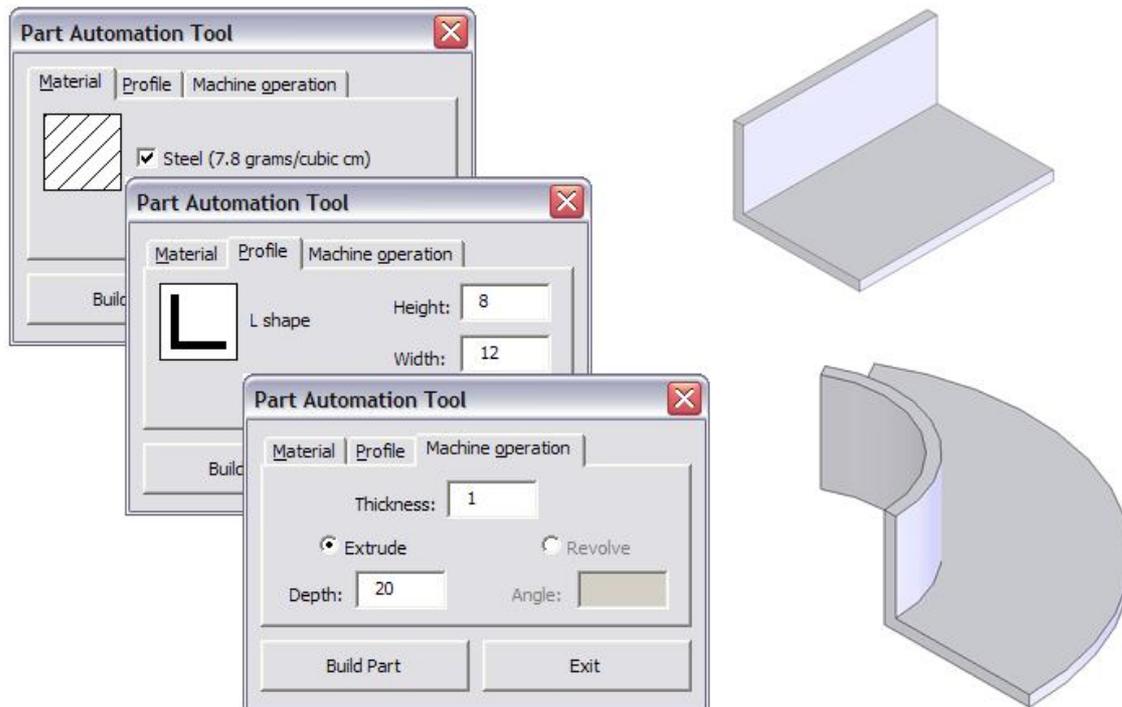
## Homework #2

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SolidWorks API

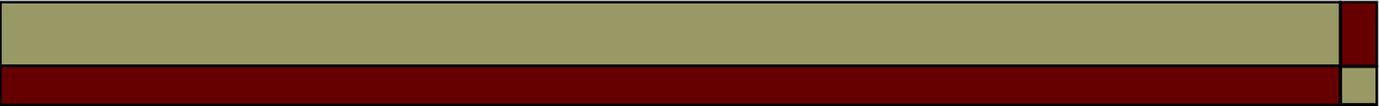
Due Date: 2009.05.05. 23:00:00

# Automating the Part Creation



## □ REQUIREMENT

- Set the material to Steel (density = 7.8 gram/cubic cm )
- Sketch a profile (L bracket) with adjustable height and width
- Extrude or revolve the profile as a thin feature with adjustable depth or angle



# Procedure & Tip

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- Procedure

1. Open the macro *AutomatingPartDesign.swp*
2. Import the SolidWorks constants file
3. Add missing controls to the form
4. Program the click event for the Build Part command button
5. Refer to the APIs listed below to automate the design
6. Save and run macro

- APIs for Reference(Recommended):

- ModelDoc2.SetUserPreferenceDoubleValue
- ModelDoc2.SetUserPreferenceStringValue
- Sldworks.SetUserPreferenceToggle
- ModelDoc2.CreateLine2
- ModelDoc2.ViewZoomtofit2
- ModelDoc2.AddDimension2
- FeatureManager.FeatureExtrusionThin
- FeatureManager.FeatureRevolveThin

# Submission details

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- Objects on demand:
  - Modified *AutomatingPartDesign.swp*
  - Make a 'README.txt' file for very short comments about the homework
  - Make a 'Zip' file name of 'HW2\_(StudentID)\_(NAME).zip' which includes the Modified *AutomatingPartDesign.swp* file and README.txt file
  
- Submit: <http://etl.snu.ac.kr/>
  - Due : May 05, 2009, 23:00:00
  - Upload to Assignment Board