

System Options

Description: Allows you to customize your work environment as well as the system performance.

Note: Accepting changes to the system options does not need to occur on every tab, it can be done after all desired changes have been made.

General

Document Settings which is only accessible when a part, assembly, or drawing is open

Choose whether to open a recent document at the startup of SolidWorks

Overall SolidWorks system and performance options

Enables the modify dialog box for changing a dimension. If this is not checked the user will have to double click a dimension to change its value

Shows the name of the dimension as well as its numeric value

Allow for a multi-user environment. The same file can be worked on at the same time as well as saved and updated between users (does not check lightweight components). Optimal for a team project setting.

Reset all toolbars to original factory settings (applies to all tabs)

Accept any changes the user has made to the general settings

System Options - General

System Options | Document Properties

General

Drawings

- Display Style
- Area Hatch/Fill

Colors

Sketch

- Relations/Snaps

Display/Selection

Performance

Assemblies

External References

Default Templates

File Locations

FeatureManager

Spin Box Increments

View

Backup/Recover

Hole Wizard/Toolbox

File Explorer

Search

Collaboration

Advanced

Open last used document(s) at startup: Never

- Input dimension value
- Single command per pick
- Show dimension names
- Show errors every rebuild
- Warn before saving documents with updates
- Maximize document on open
- Use shaded face highlighting
- Show thumbnail graphics in Windows Explorer
- Use system separator for dimensions
- Use English language menus
- Use English language feature and file names
- Enable Confirmation Corner
- Auto-show PropertyManager
- Automatically edit macro after recording
- Enable FeatureXpert

When rebuild error occurs: Prompt

Custom property used as component description: Description

- Show latest news feeds in task pane
- Enable performance feedback

(see Customer Experience Feedback Program)

- Disable additional weldment configuration description strings

Reset All

OK Cancel Help

Drawings

The screenshot shows the 'System Options - Drawings' dialog box. The 'Drawings' category is selected in the left-hand tree. The right-hand pane contains various checkboxes and settings. Three red arrows point from callout boxes to specific settings:

- The first arrow points to the 'Show contents while dragging drawing view' checkbox, which is checked. The callout box says: "Displays a shadow box outline of the drawing view when inserted onto a sheet".
- The second arrow points to the 'Display sketch arc centerpoints' checkbox, which is checked. The callout box says: "Enable the display of sketch arc center points when a drawing view is inserted".
- The third arrow points to the 'Detail view scaling' field, which is set to '2'. The callout box says: "Set the automatic scale for a detail view".

Other visible settings include: 'Eliminate duplicate model dimensions on insert' (checked), 'Mark all part/assembly dimension for import into drawings by default' (checked), 'Automatically scale new drawing views' (checked), 'Smooth dynamic motion of drawing views' (checked), 'Print out-of-date drawing views with crosshatch' (set to 'Prompt'), and 'Keyboard movement increment' (set to '10mm').

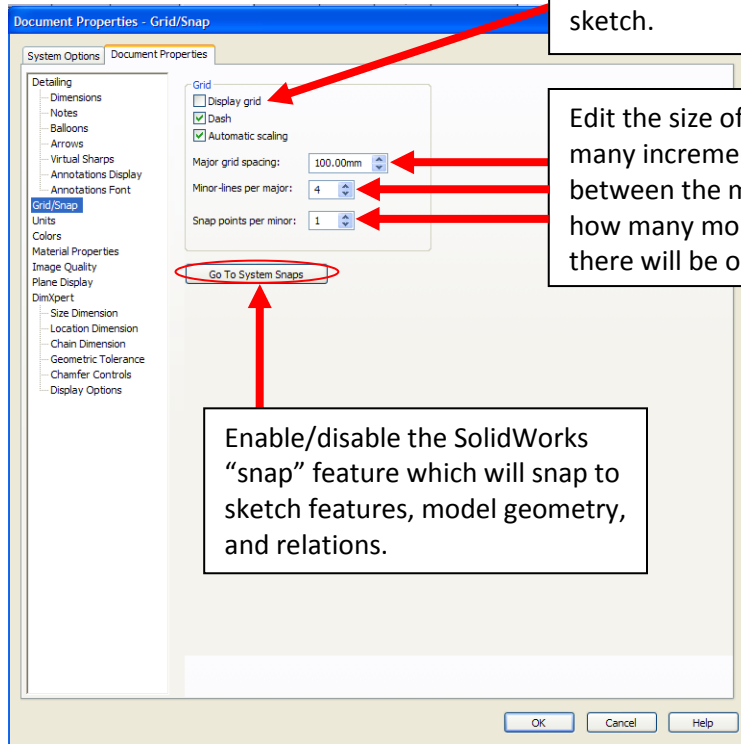
Detailing

The screenshot shows the 'Document Properties - Detailing' dialog box. The 'Detailing' category is selected in the left-hand tree. The right-hand pane contains settings for dimensioning and datum features. Two red arrows point from callout boxes to specific settings:

- The first arrow points to the 'Dimensioning standard' dropdown menu, which is set to 'ISO'. The callout box says: "Change the dimensioning standard (ANSI → ISO)".
- The second arrow points to the 'Datum features' section, specifically the 'Display type' dropdown menu, which is set to 'Per Standard'. The callout box says: "Edit datum features such as which letter it begins at and the shape of the surrounding text box".

Other visible settings include: 'Extension lines' (Gap: 1mm, Beyond dimension line: 1mm), 'Datum features' (Next label: A), and 'Surface finish symbols' (Display symbols per 2002: unchecked).

Grid/Snaps



Display a grid when entered in a sketch.

Edit the size of the grid, how many incremental lines are between the major lines, and how many mouse snap points there will be on each minor line

Enable/disable the SolidWorks "snap" feature which will snap to sketch features, model geometry, and relations.

Basic Units

Specify the units if custom unit systems is selected. Specify the precision as either decimal or fractions under fractions. Under more you can round to the nearest fraction. Specify the units for angles (deg/rad).

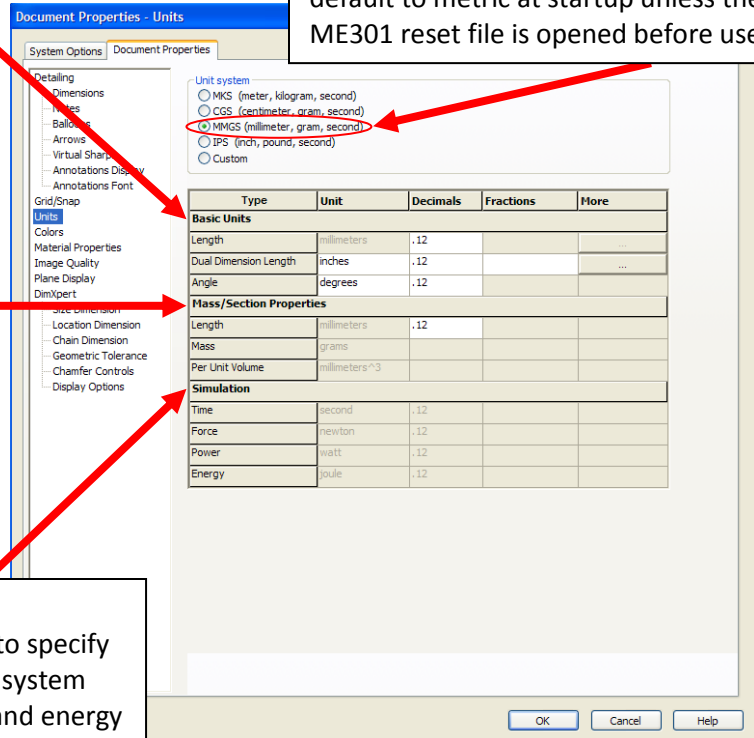
Mass/Selection Properties

Specify the length, mass, and per unit volume units. The mass properties use these values to calculate part properties such as center of mass and moment of inertia.

Simulation

This can only be used with Assemblies, to specify the decimal precision. In a custom unit system can specify units of time, force, power and energy for special applications where the default units would not work.

Units



Set the desired units, the two choices are SI and metric. SolidWorks will default to metric at startup unless the ME301 reset file is opened before use.