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in Analysis tab of action
bar, click on sketch

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analysis icon. A panel will appear, it will tell you. if the sketch is constrained, under constrained or over constrained; if it is opened or closed; the list of all the elements in your sketch (points, lines, arc, splines, etc.) Since the sketch is opened, click on close opened profile feature!

Warning ! this will close

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Two simple ways are by using the top pull down menu (Start – Mechanical Design – Sketcher), or by selecting the Sketcher icon. When you enter the sketcher, CATIA requires that you choose a plane to sketch on. You can choose this plane either before or

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