



Document - CS37254

Document Information

Type	Article
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Document Details

Title	How to configure Expert Moldbase Extension (EMX) in Creo Parametric, Pro/ENGINEER or Creo Elements/Pro 5.0
Description	<ul style="list-style-type: none"> How to configure Expert Moldbase Extension in Creo Parametric How to configure Expert Moldbase Extension in Pro/ENGINEER How to configure Expert Moldbase Extension in Creo Elements/Pro 5.0
Applies To	<ul style="list-style-type: none"> Expert Moldbase Extension 6.0 - 9.0 all datecodes Pro/ENGINEER Wildfire 2.0 and later all datecodes Creo Elements/Pro 5.0 all datecodes Creo Parametric 1.0 - 3.0 all datecodes
Cause	
Resolution	<ul style="list-style-type: none"> To configure EMX after Pro/ENGINEER launches <ul style="list-style-type: none"> For Creo Parametric <ul style="list-style-type: none"> Click Home tab > Utilities > Auxiliary Applications > Register... > browse to the protk.dat file > Open > select the item in Auxiliary Applications dialogue box > Start > Close <ul style="list-style-type: none"> Location of protk.dat file: <EMX_loadpoint>\bin\<Pro/ENGINEER release> For Pro/ENGINEER Wildfire 2.0-5.0 and Creo Elements/Pro 5.0 <ul style="list-style-type: none"> Click Tools > Auxiliary Applications > Register... > browse to the protk.dat file > Open > select the item in Auxiliary Applications dialogue box > Start > Close To configure EMX at the same time when Pro/ENGINEER launches, there are two methods as below: <ul style="list-style-type: none"> Set config.pro option protkdat to the full path of protk.dat file <ul style="list-style-type: none"> About config.pro file location: <ul style="list-style-type: none"> For Creo Parametric, File > Options > Configuration Editor For Pro/ENGINEER Wildfire 2.0-5.0 and Creo Elements/Pro 5.0, Tools > Options For example, protkdat <EMX_loadpoint>\bin\<Pro/ENGINEER release>\protk.dat Copy protk.dat file to the Start in directory of Pro/ENGINEER

SPR Information

Related Documents

Type	Number	Status	Description
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